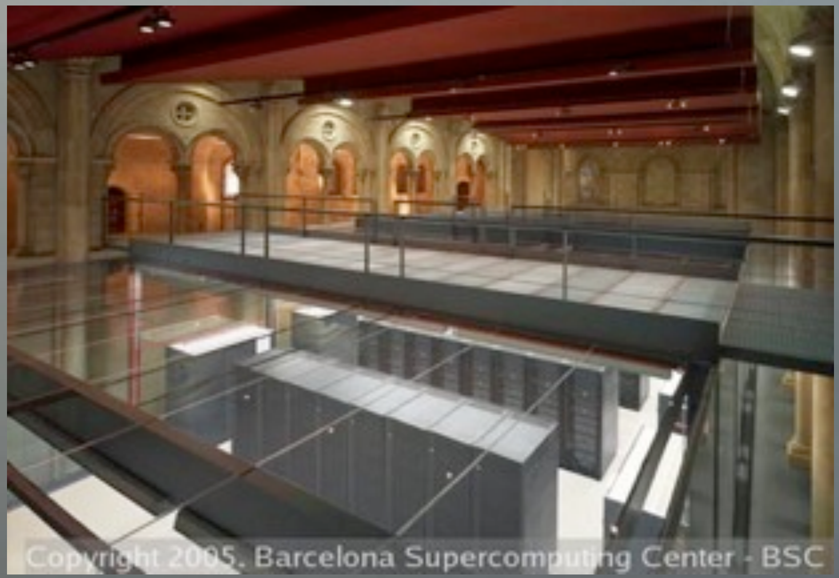
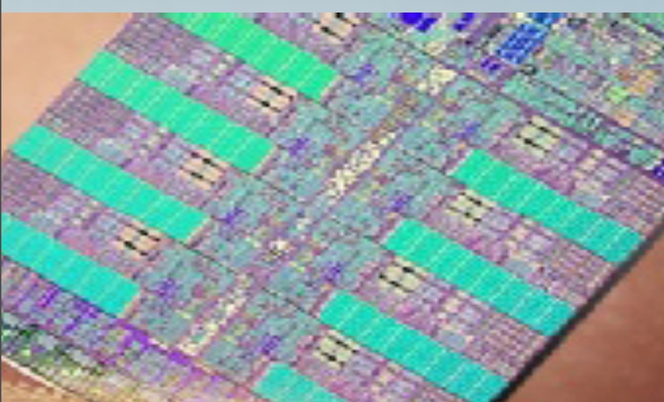
High Performance Computational Fluid Dynamics at BSC -Driving & Sailing

CASE Department
Barcelona Supercomputing Center - Centro Nacional de Supercomputación (BSC-CNS)
Spain

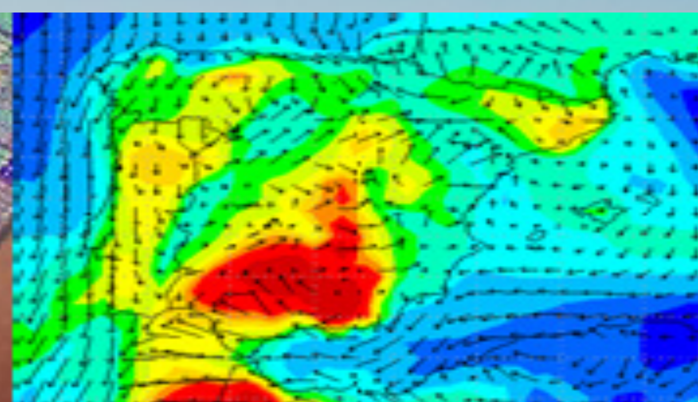
Contact:
mariano.vazquez@bsc.es
herbert.owen@bsc.es



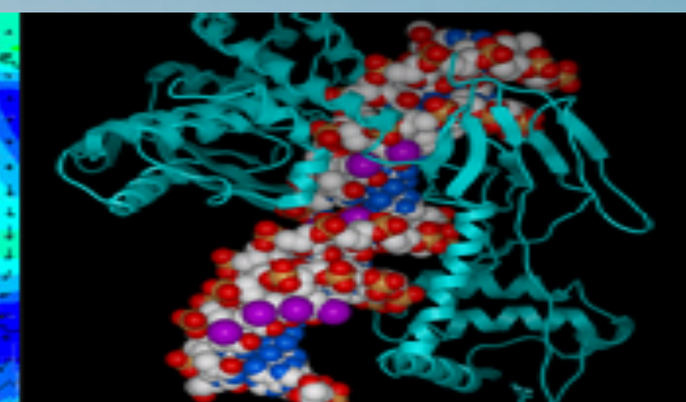
BSC-CNS departments



COMPUTER SCIENCES



EARTH SCIENCES



LIFE SCIENCES

COMPUTER APPLICATIONS

Computational Mechanics

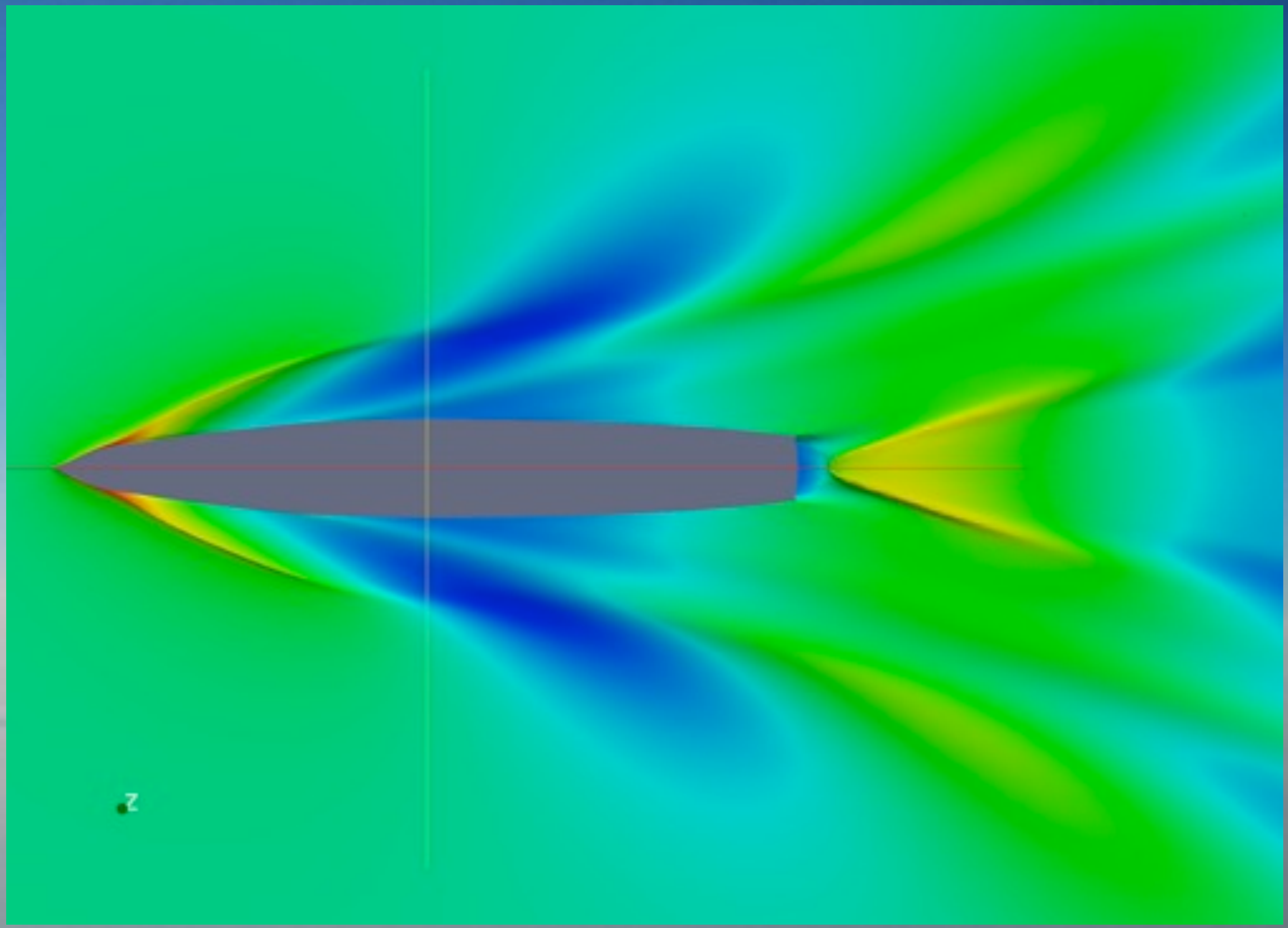
Applied Computer Science

Optimization

CASE Department - Objectives

Computer Applications in Science

and Engineering (CASE) Department



Interdisciplinary research unit of the BSC-CNS

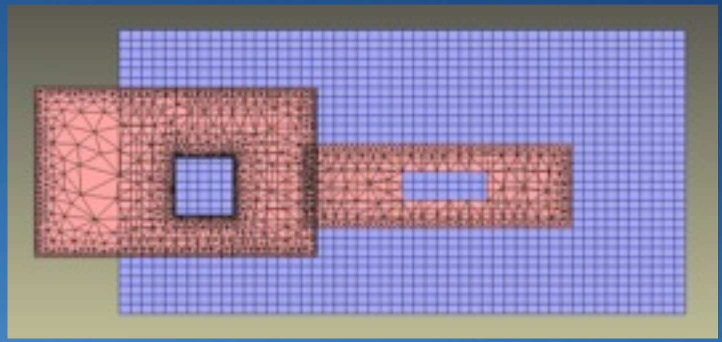
Our mission:

To develop **computational tools** to simulate **highly complex** problems

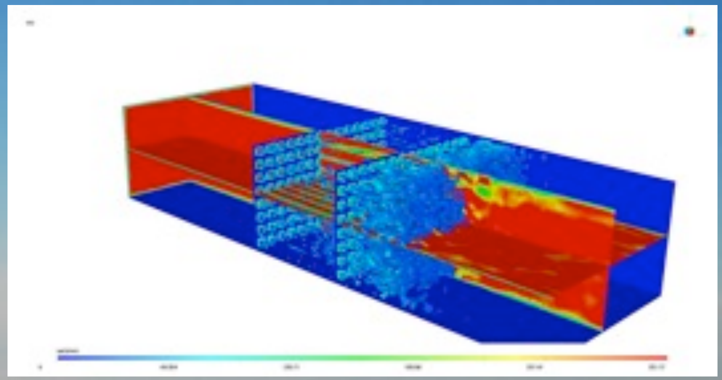
seamlessly adapted to run onto **high-end parallel supercomputers**

CASE Department – Research

Research lines



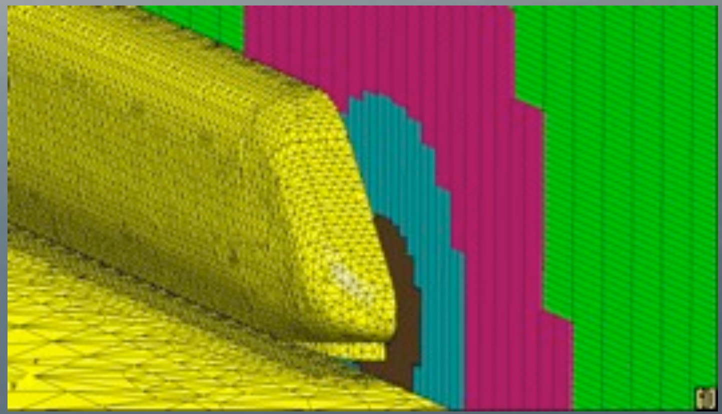
Physical and Numerical Modeling



Mesh Generation and Numerical Solution Algorithms

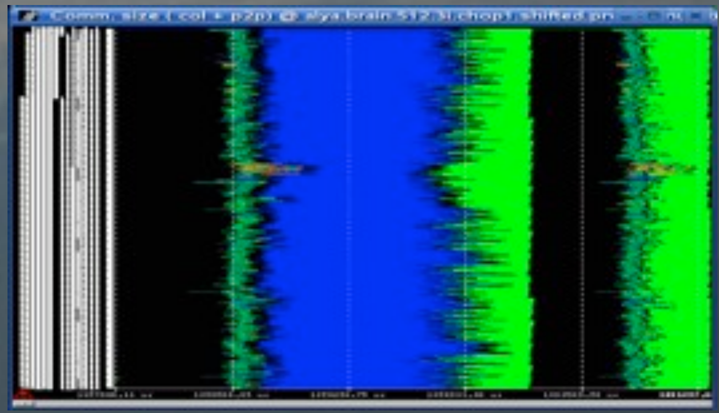
High Performance Computing in Computational Mechanics

Parallelization in Distributed and Shared memory machines



Optimization

Algebraic Solvers



CASE Department – Research

Application lines

Bio Mechanics

Meteorology

Building, Energy & Environment

Aerospace Industry

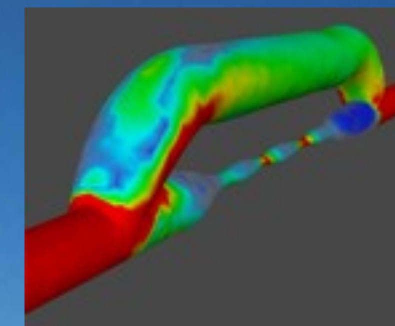
Land Vehicles Industry

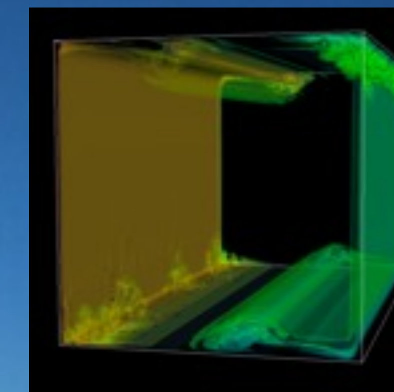
Ship Hydrodynamics

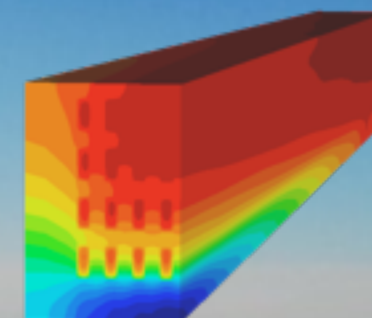
Oil Industry

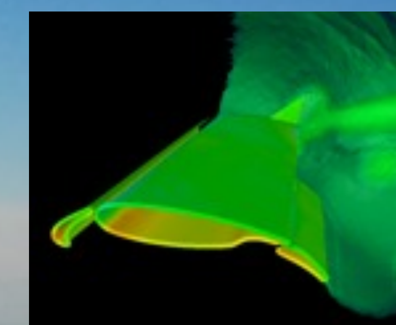
Material Science

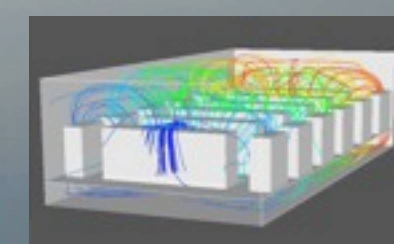
Artificial Societies

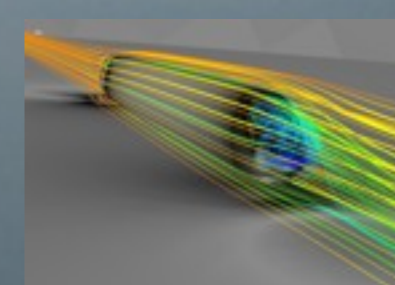


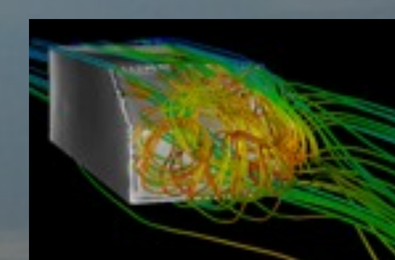


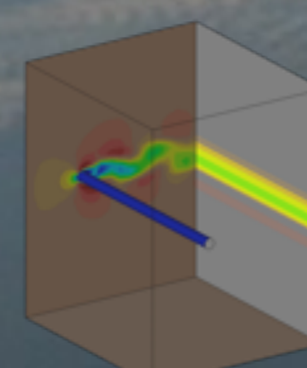


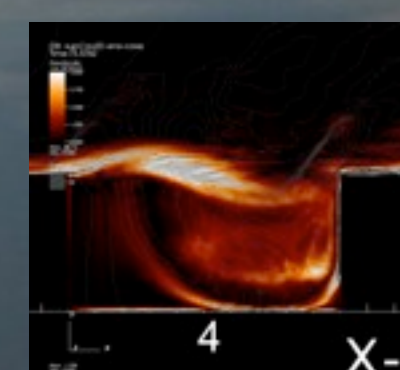




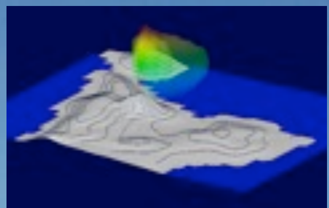


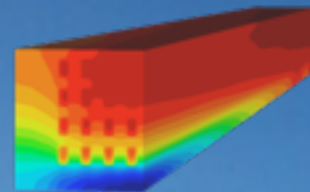
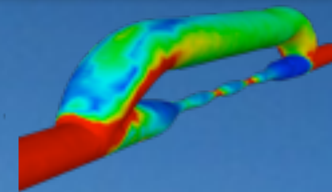
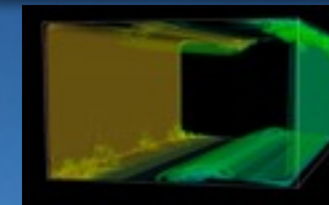
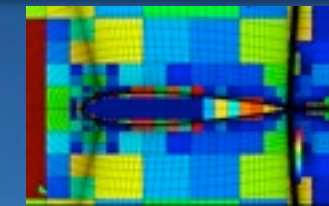
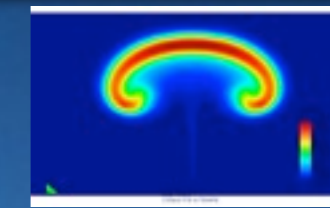






ALYA System: High Performance Computational Mechanics



BSC In-house code

Multi-physics modular code for High Performance Computational Mechanics & Design

Numerical solution of PDE's

Variational methods are preferred (FEM)

Hybrid meshes, non-conforming meshes

Explicit and Implicit formulations

Coupling between multi-physics (loose or strong)

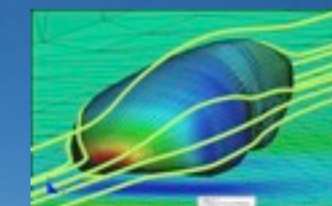
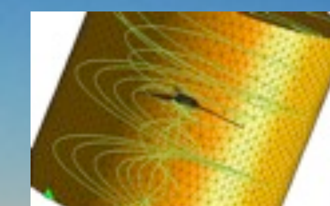
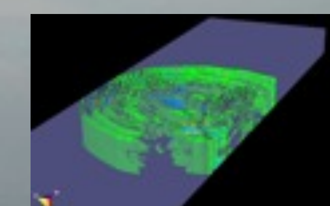
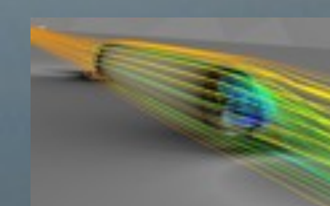
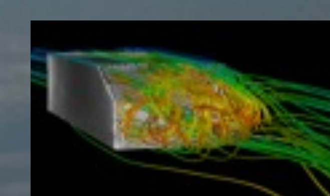
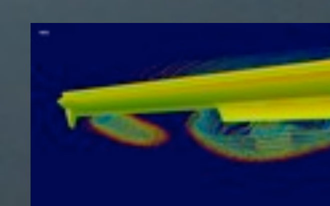
Advanced meshing issues

Parallelization by MPI and OpenMP

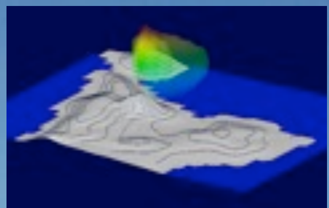
Automatic mesh partition using Metis

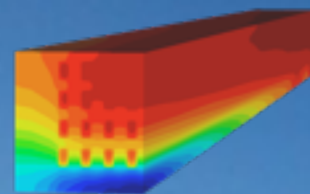
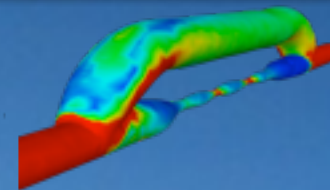
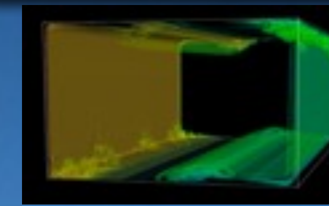
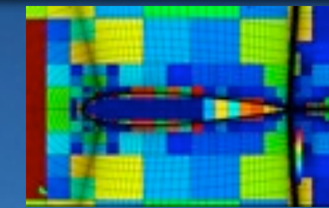
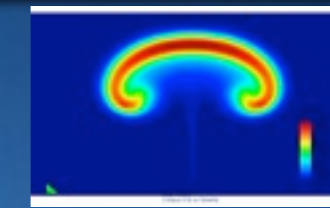
Portability is a must

Porting to new architectures: CELL, GPUs, ...

ALYA System: High Performance Computational Mechanics



Compressible / Incompressible Navier-Stokes

Turbulence (RANS & LES models)

Free surface

Non-linear solid mechanics

Wave equation

Heat transport

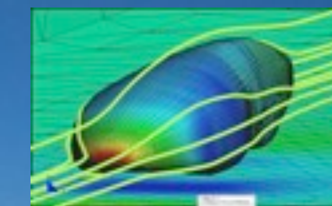
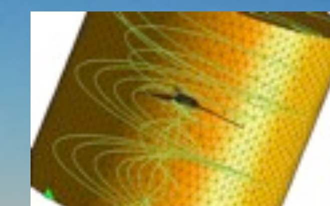
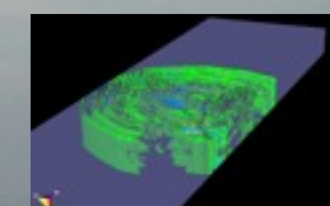
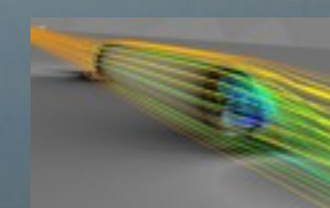
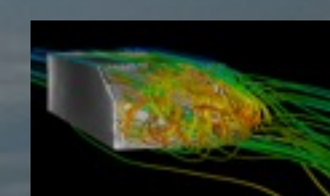
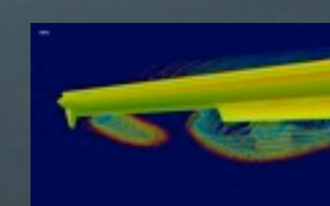
Excitable media

Helmholz equation

Particle transport

Chemical reactions

Many bodies

RANS: Reynolds Average Navier Stokes

Obtains a stationary solution that is representative of the mean flow.

Models the effect of the fluctuating flow on the mean flow.

Very limited application to transient simulations.

Much cheaper computationally. At least 1 order of magnitude.

Wide spread use in the industry. Available in all commercial codes.

More robust. Easier to use.

Poor results when there are important separated regions.

Very limited application to transient simulations such as Aeroacoustics.

Some models: Spalart Allmaras ; SST k-omega ; k-epsilon

LES: Large Eddy Simulation

Transient simulations: at least 2nd order time accurate Methods.

Also higher accuracy (less diffusive numerical schemes) in space.

The evolution of the Large Eddies is simulated.

Must be averaged over large time periods of time.

Much higher computational cost. Highly scalable and efficient codes required.

Can not be used to model the flow close to the wall for High Reynolds problems due to computational cost. (Al least for the next 2 decades).

Wall Laws for LES.

Hybrid RANS / LES. Detached Eddy Simulation - DES

Some models: Smagorinsky ; WALE ; Sigma ; VMS (MILES)

Discretization and Numerical method

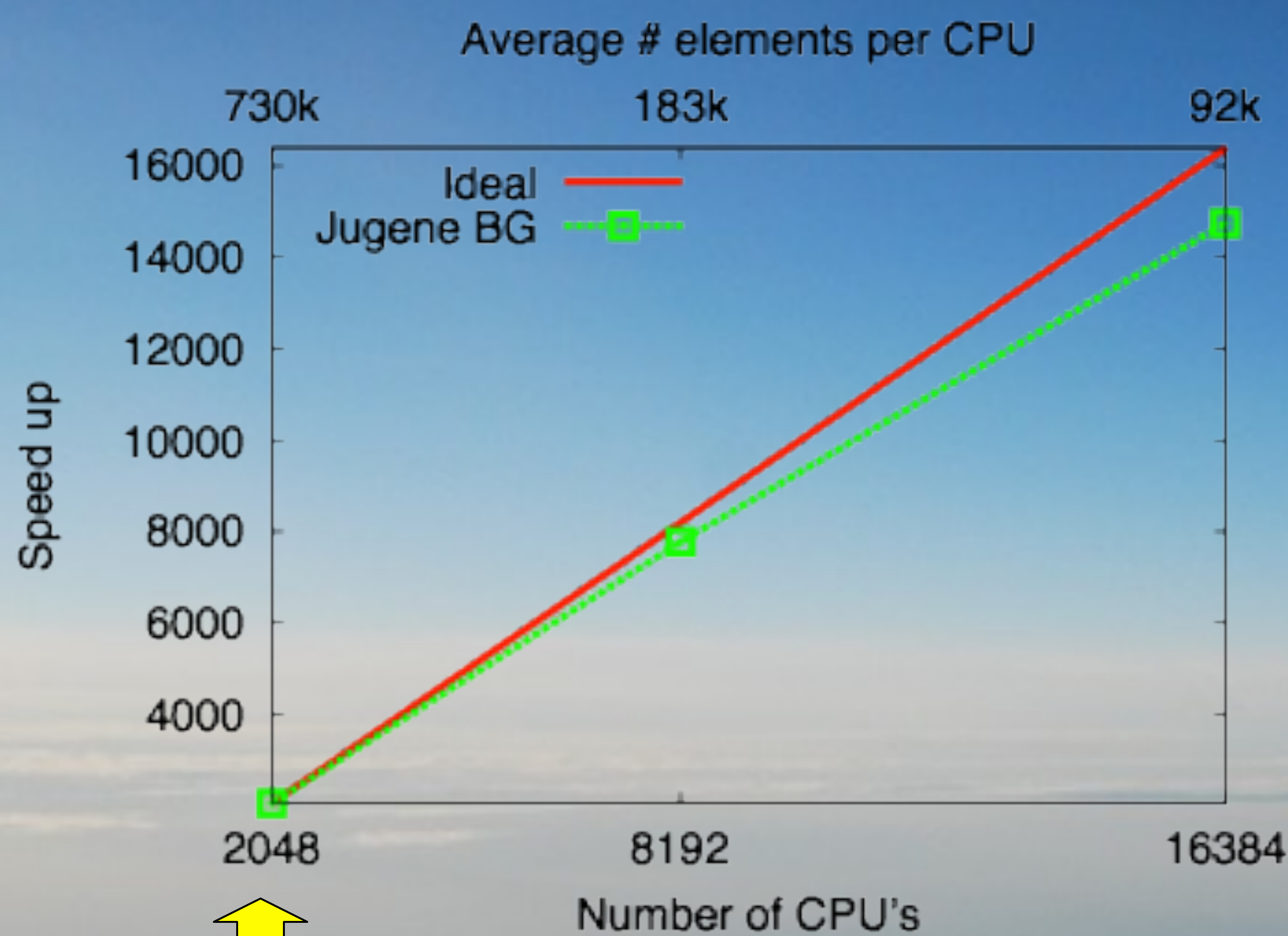
- VMS – ASGS – Stabilized finite element formulation.
- Unstructured Tetrahedras, Prisms and Pyramids.

Flow Algorithm

- Ortomin(1) solver for the Pressure Schur complement.

It is an extension of fractional step or SIMPLE techniques for incompressible flows where the relaxation parameter is obtained dynamically from the Orthomin algorithm.

- GMRES for velocity + Deflated CG for pressure.



1 hour to less than 2 seconds with no added cost!!!!

Benchmark

Aneurism geometry provided by R. Cebral, GMU, USA

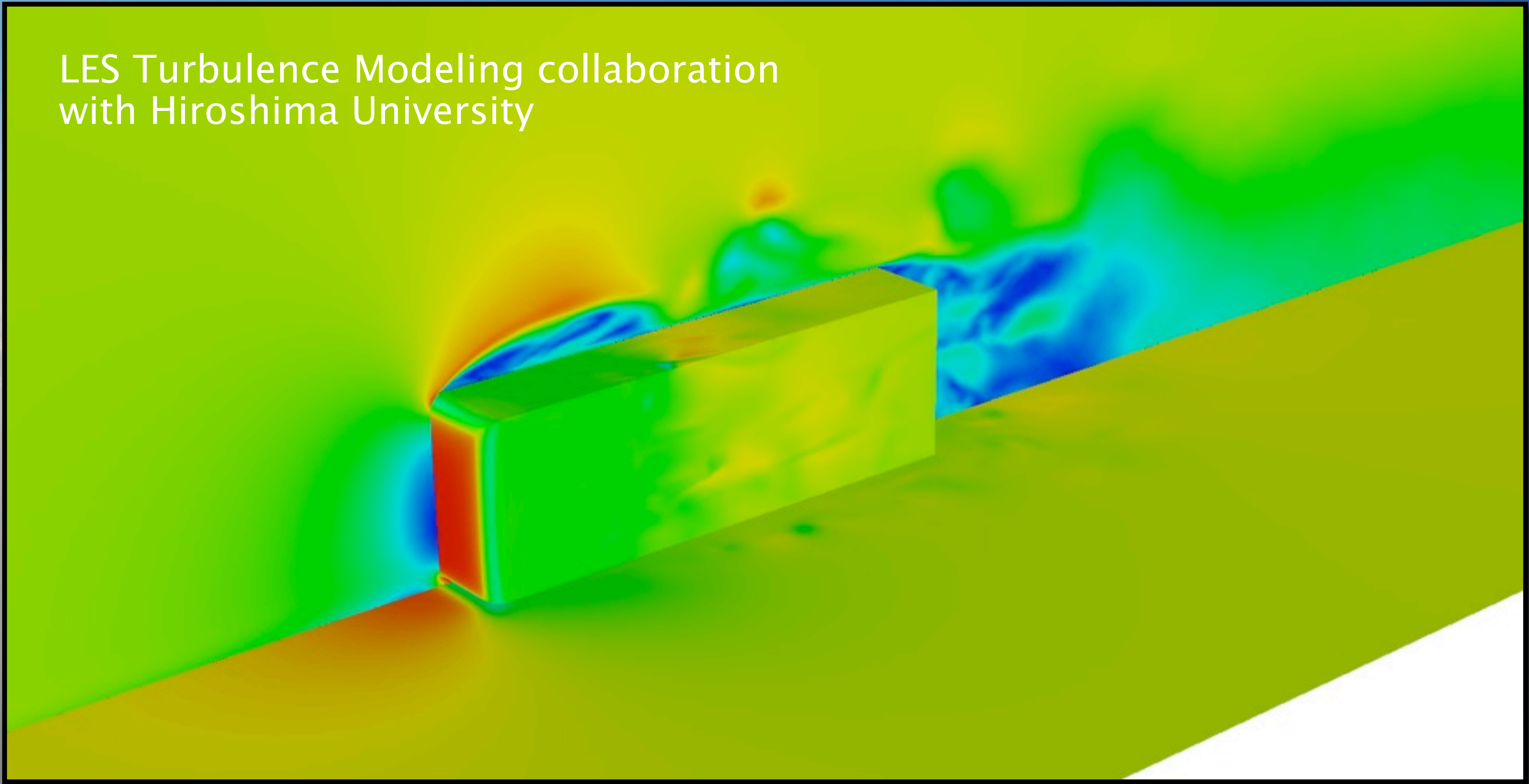
Uniform refinement up to 1.6 Billion tetrahedra

Incompressible flow

Implicit formulation

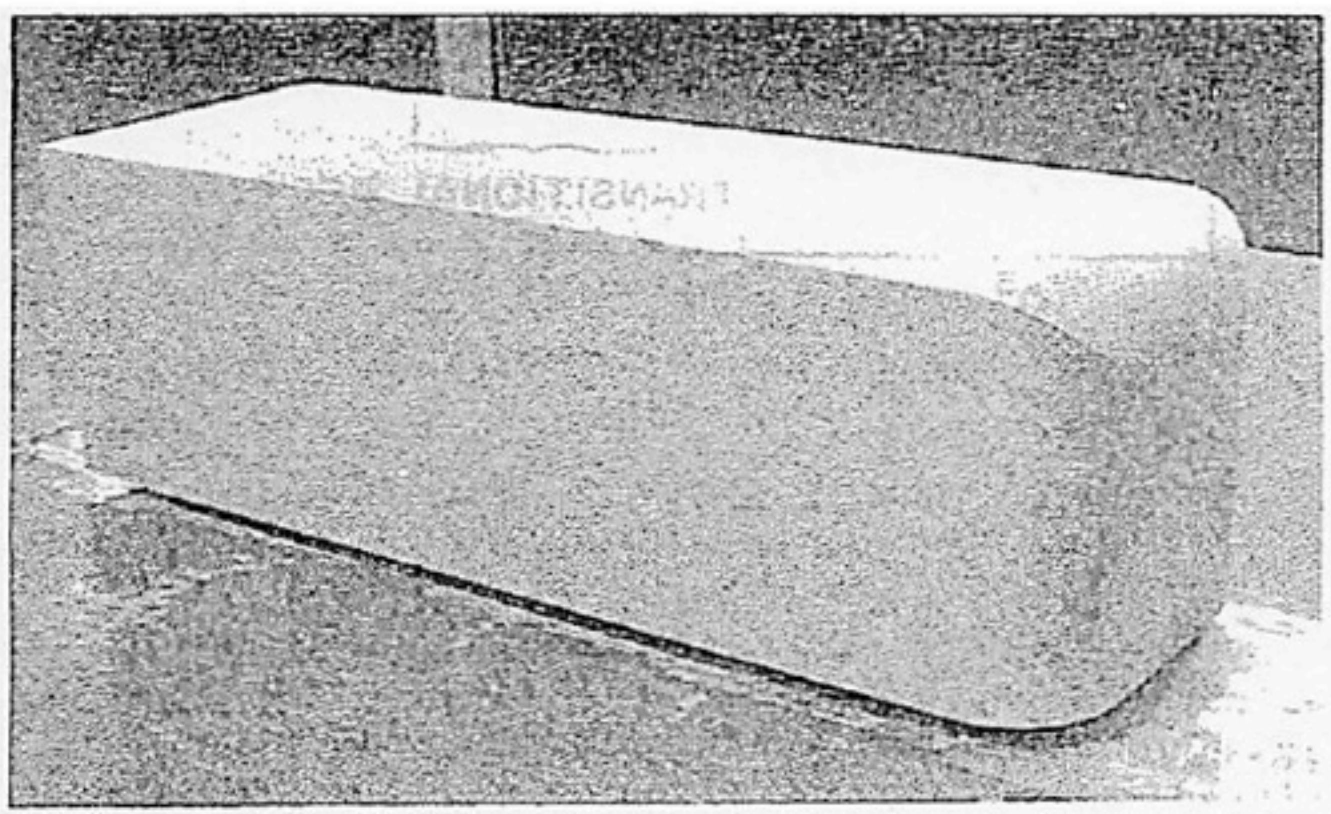
Algebraic Fractional Step: BCGStab + Deflated CG

Vehicle Aerodynamics



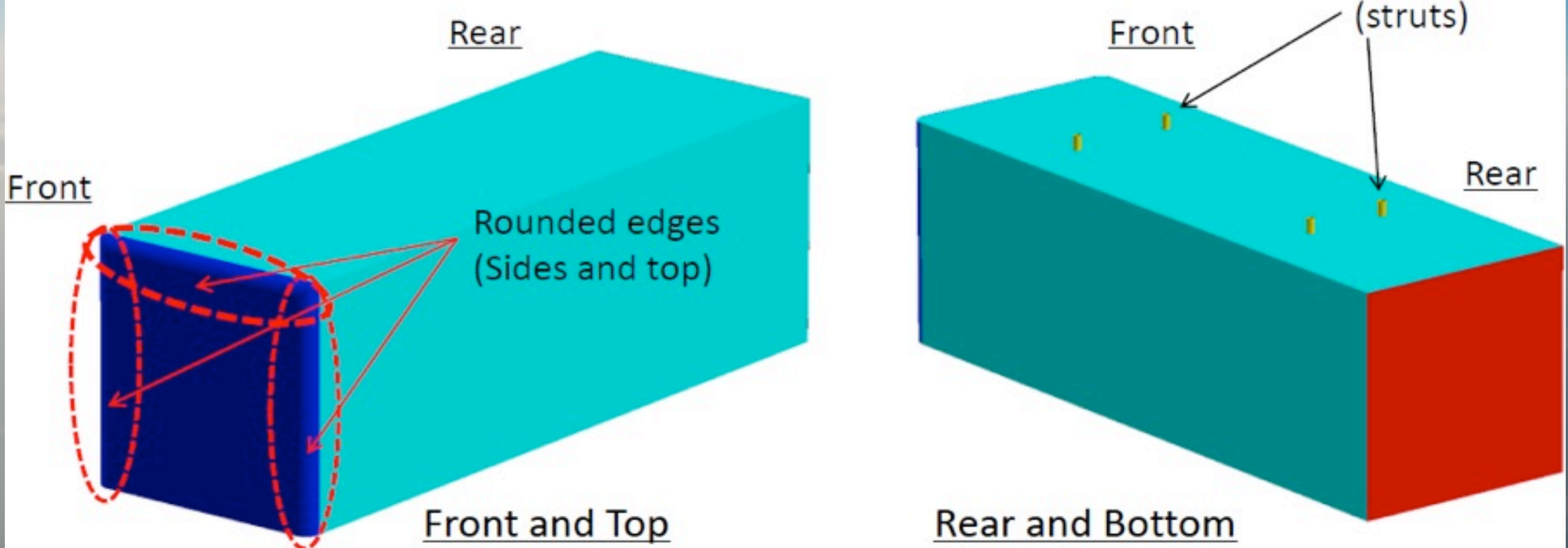
Reference paper

- Cooper (1985)
  - "The Effect of Front-Edge Rounding and Rear-Edge Shaping on the Aerodynamic Drag of Bluff Vehicles in Ground Proximity", SAE paper no. 850288, 1985.



Objective model 1/2

- Box with front-edge rounding
  - Schematic views

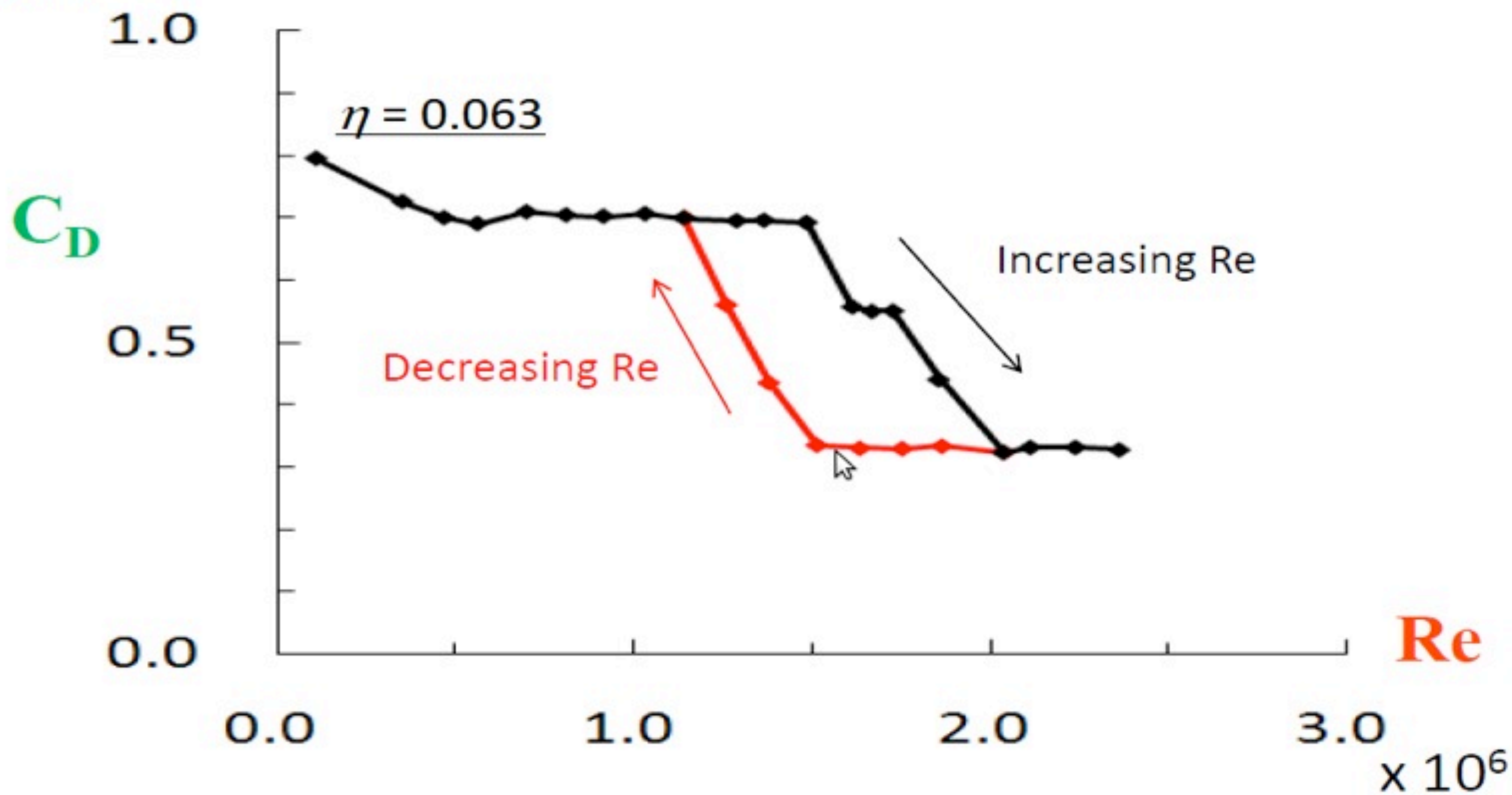


  - Size: Length L=1143mm, Width w=381mm, Height h=381mm
  - *Curvature radius of the rounded front edge:* η

Interesting phenomenon

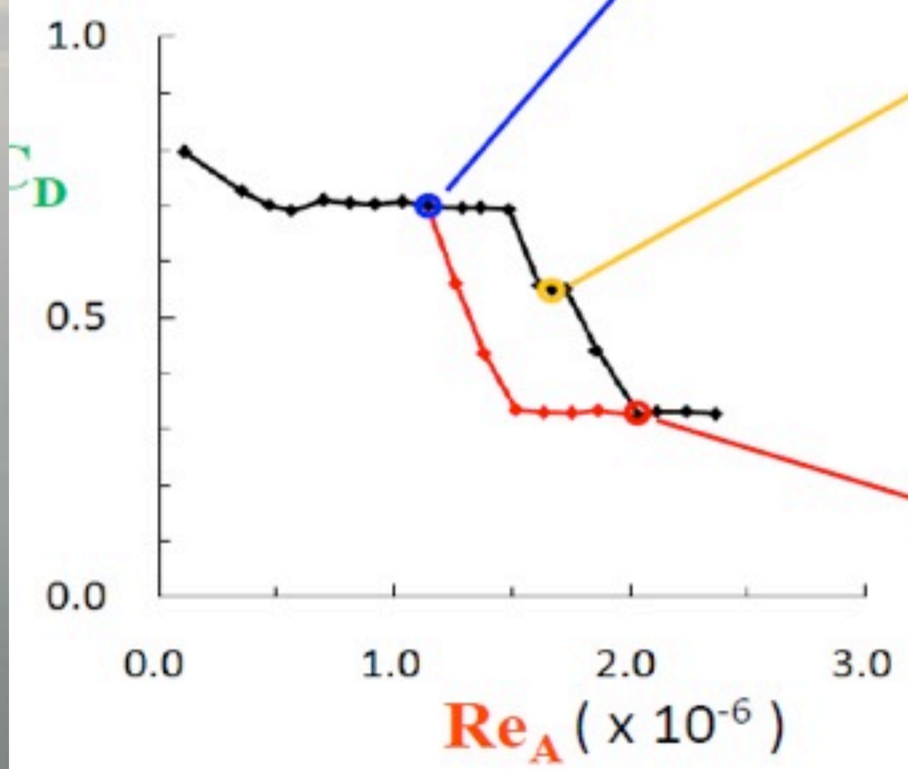
Hysteresis of aerodynamics

- Cooper's result
  - Digitized



Surface flow in Experiment (Cooper, 1985)

THREE different flow structures around the body



$Re_A = 1.29 \times 10^6$ (U=50m/s)
$C_D = 0.695$
Flow Separation
Top: Yes, Side: Yes

$Re_A = 1.67 \times 10^6$ (U=60m/s)
$C_D = 0.550$
Flow Separation
Top: Yes, Side: No

$Re_A = 2.04 \times 10^6$ (U=80m/s)
$C_D = 0.324$
Flow Separation
Top: No, Side: No

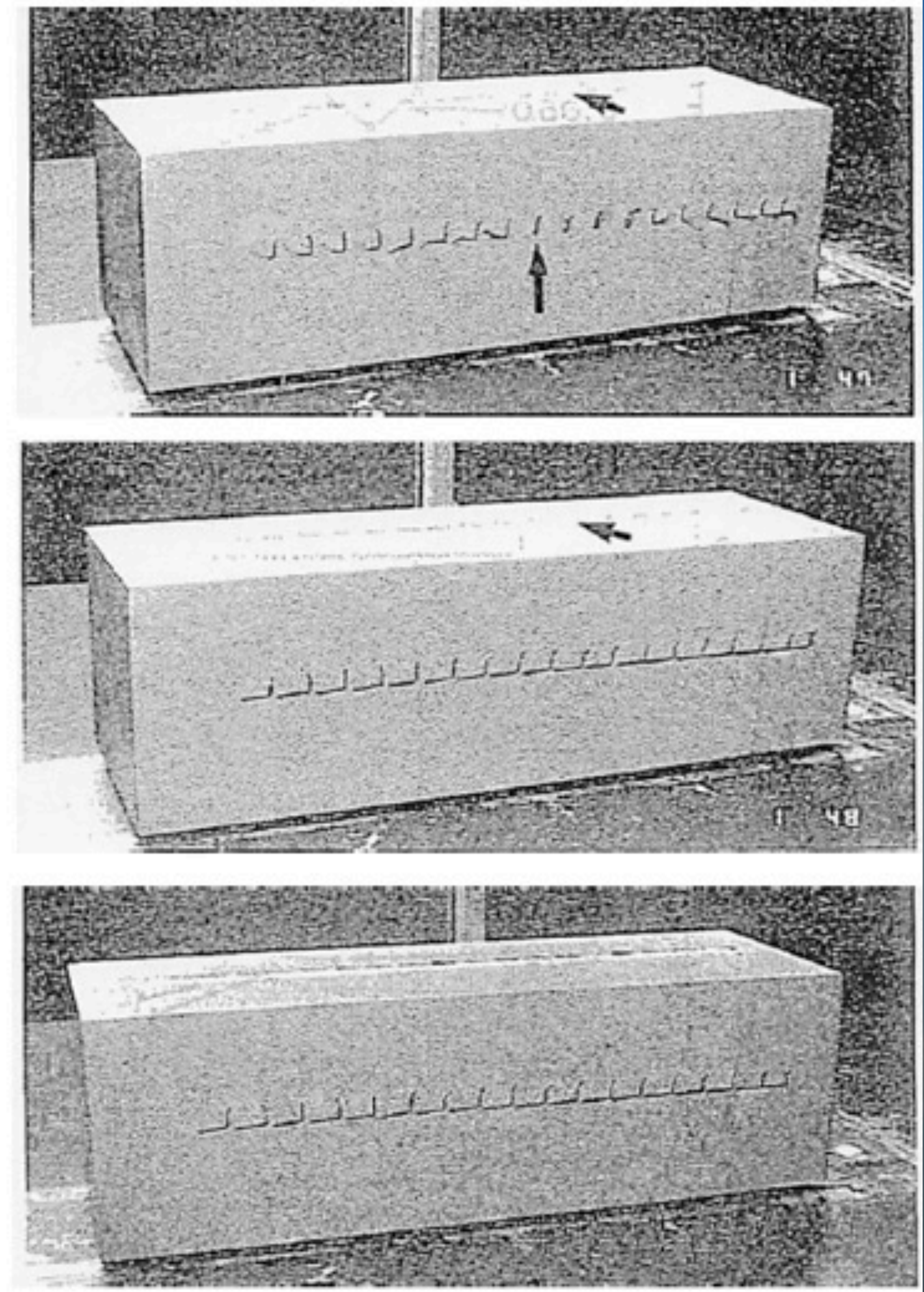


FIG. 16: SURFACE FLOW AT $\psi = 0°$ WITH $\eta = 0.063$
(REATTACHMENT POINTS INDICATED BY ARROWS)

NUMERICAL AND PHYSICAL MODEL

•Wall-adapting local eddy-viscosity (WALE) model

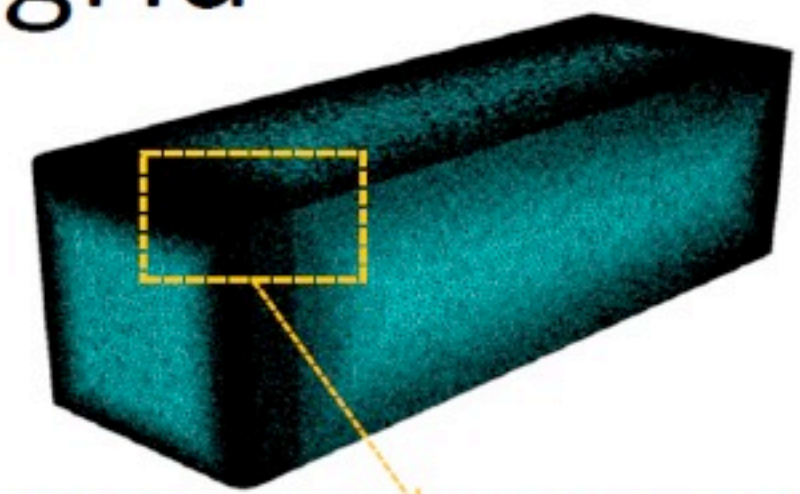
• Stabilized Finite element method with tracking of sub-scales (evolution of SUPG and GLS techniques).

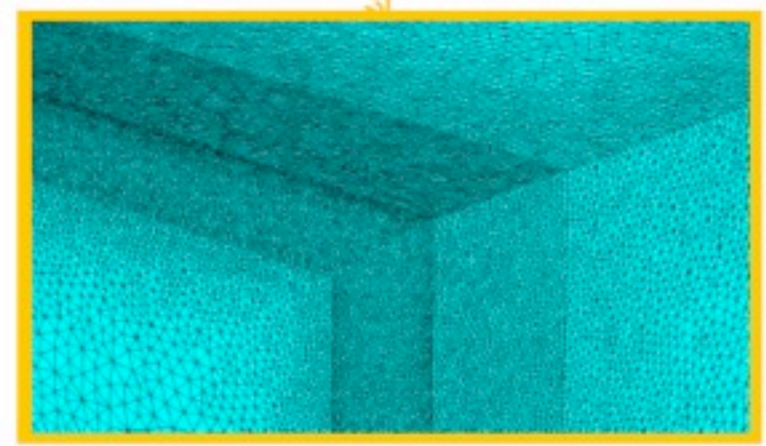
• Collaboration with professor Takuji Nakashima from Hiroshima University (Japan).

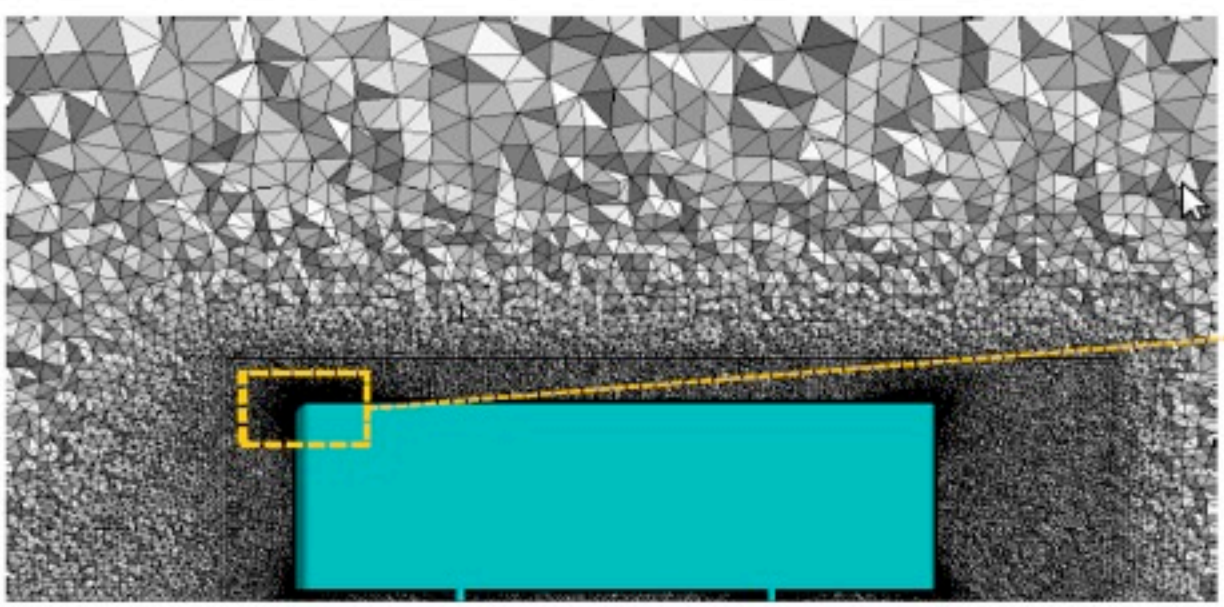
Computational grid

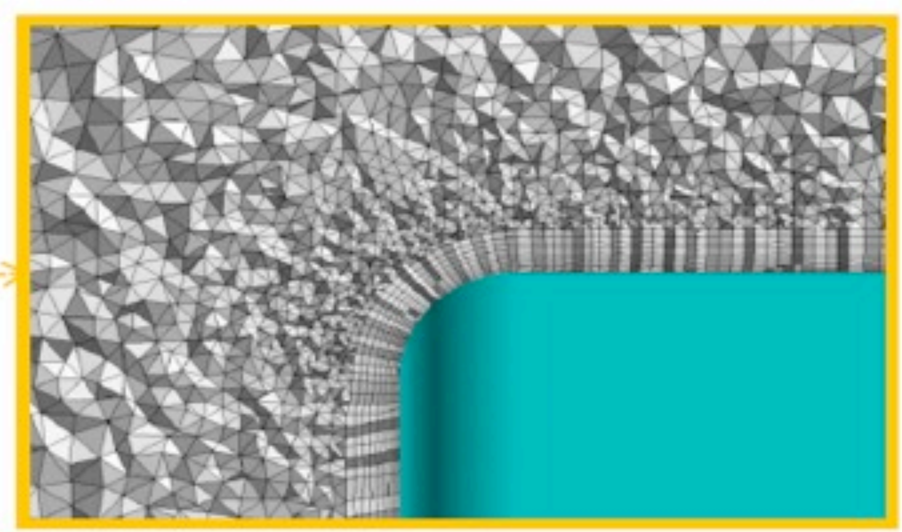
- Unstructured grid

NODAL_POINTS= 6409847
ELEMENTS= 20387394



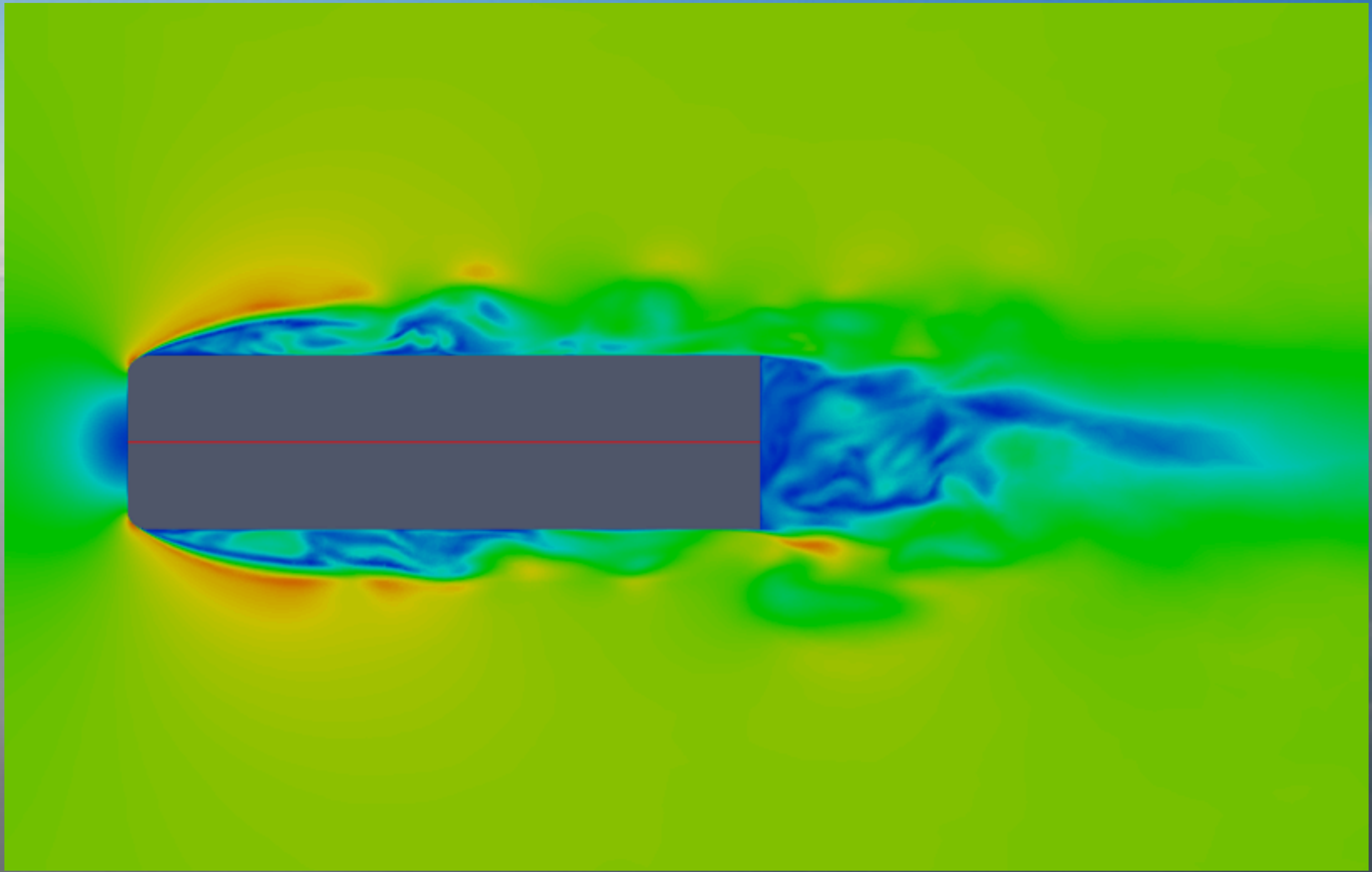






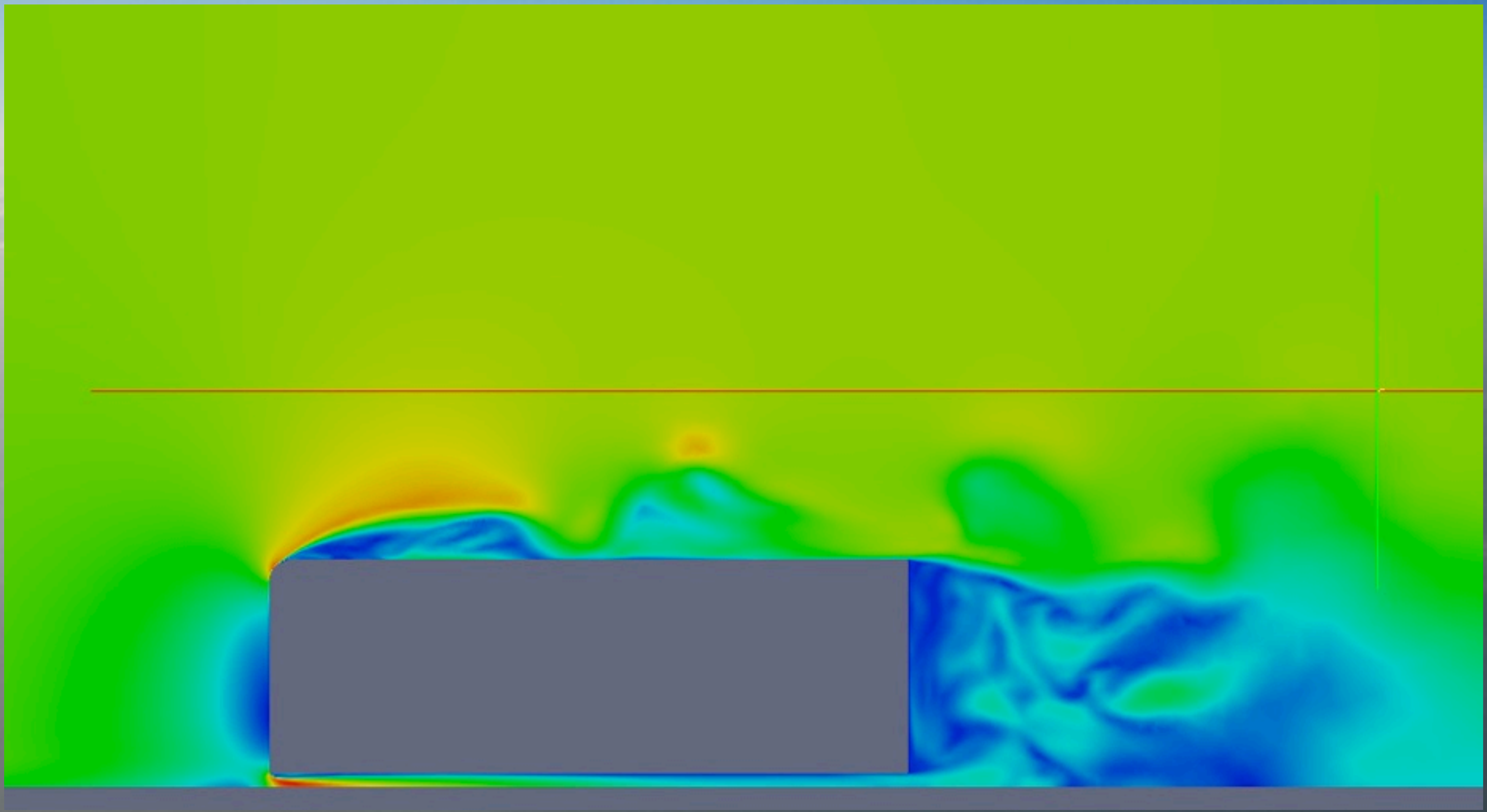
Numerical results

Velocity Top view



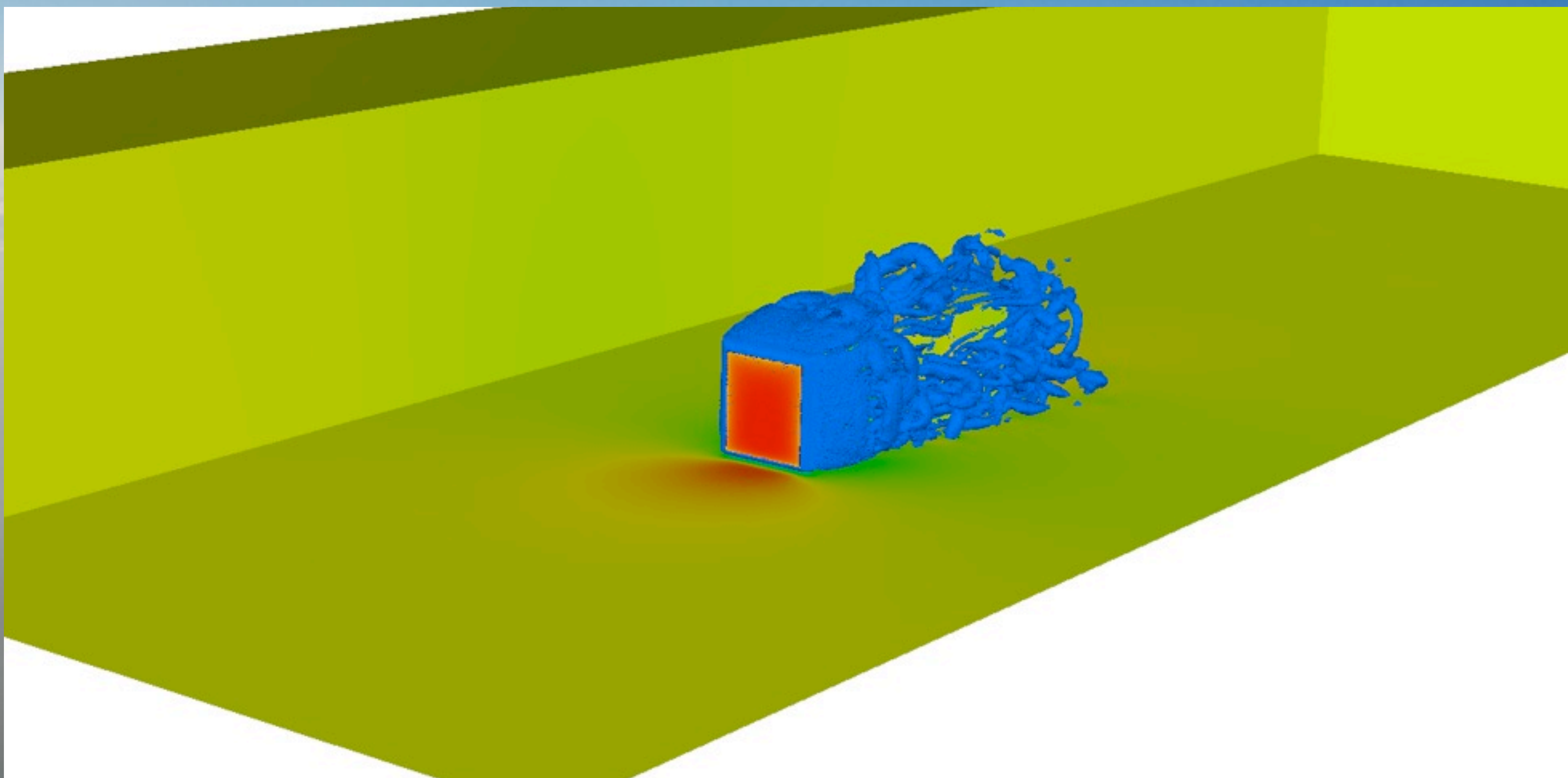
Numerical results

Velocity Lateral view

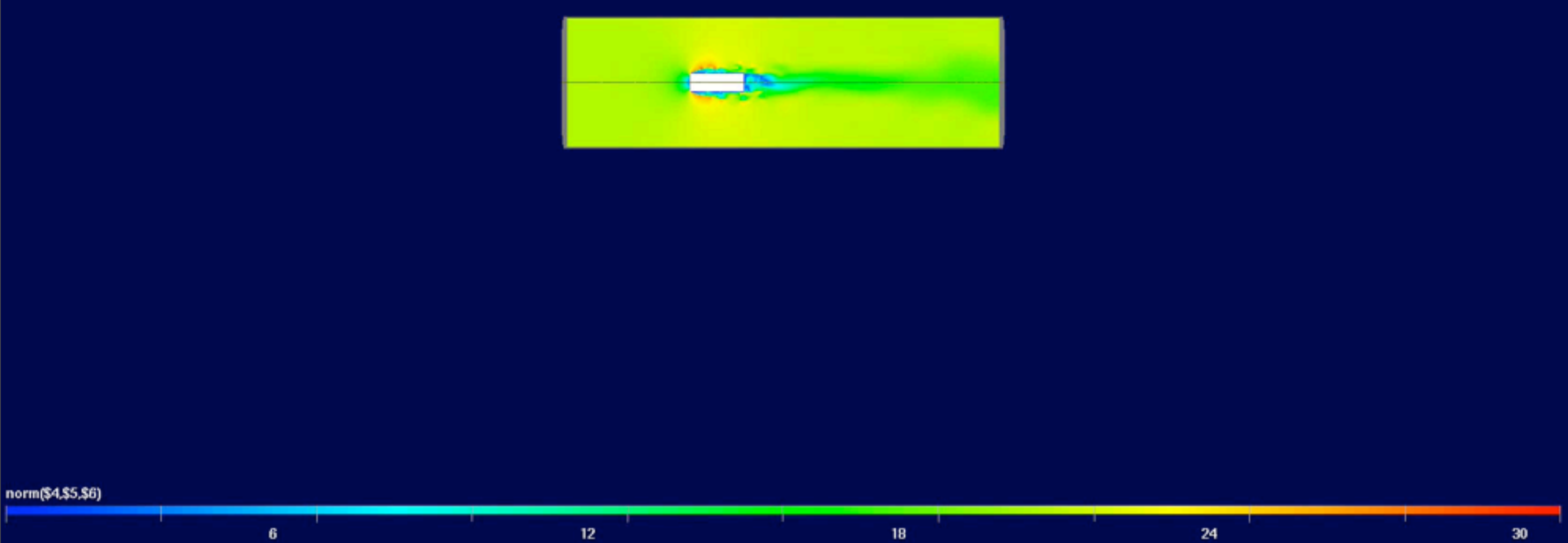


Numerical results

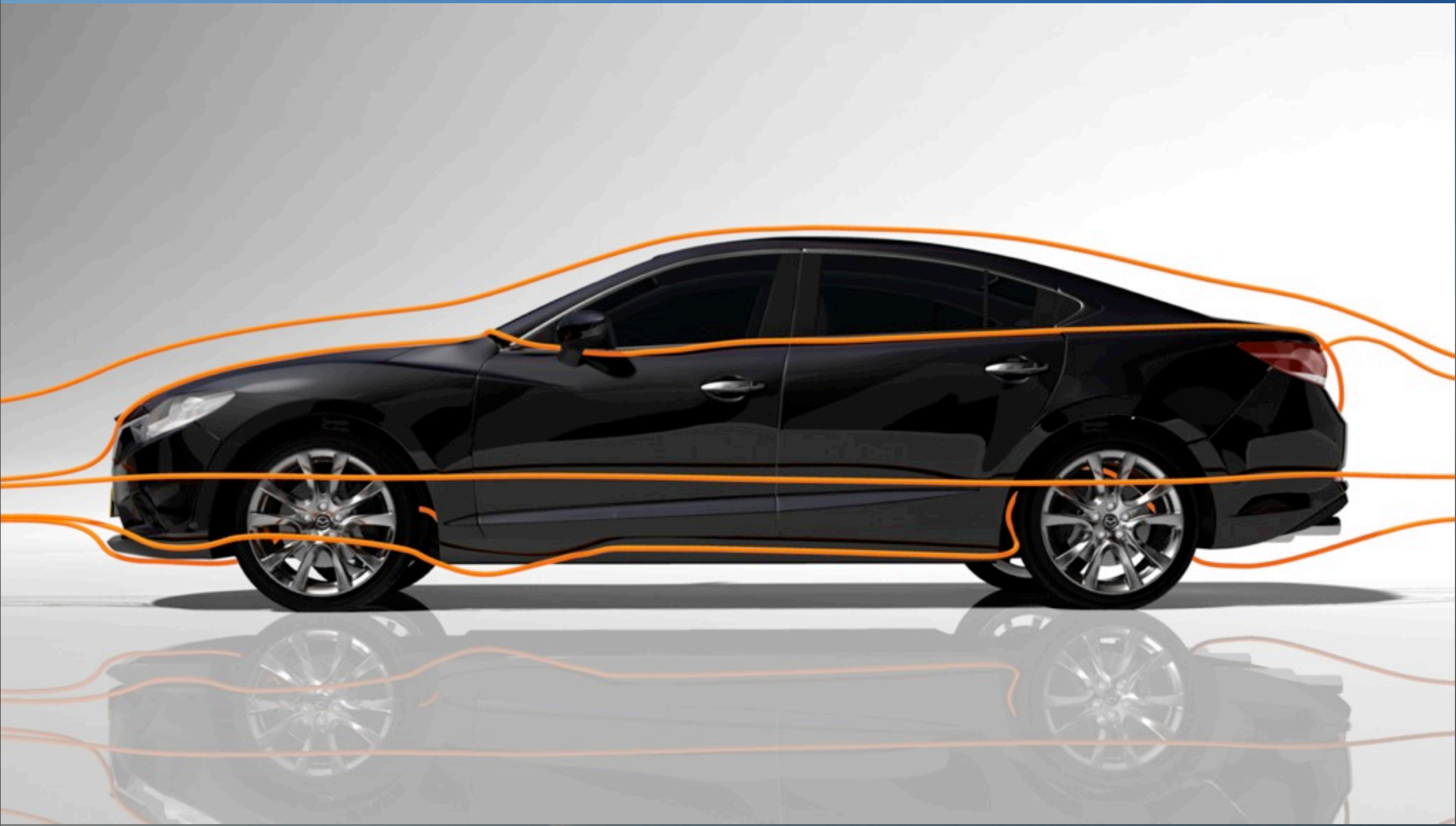
Qvorticity



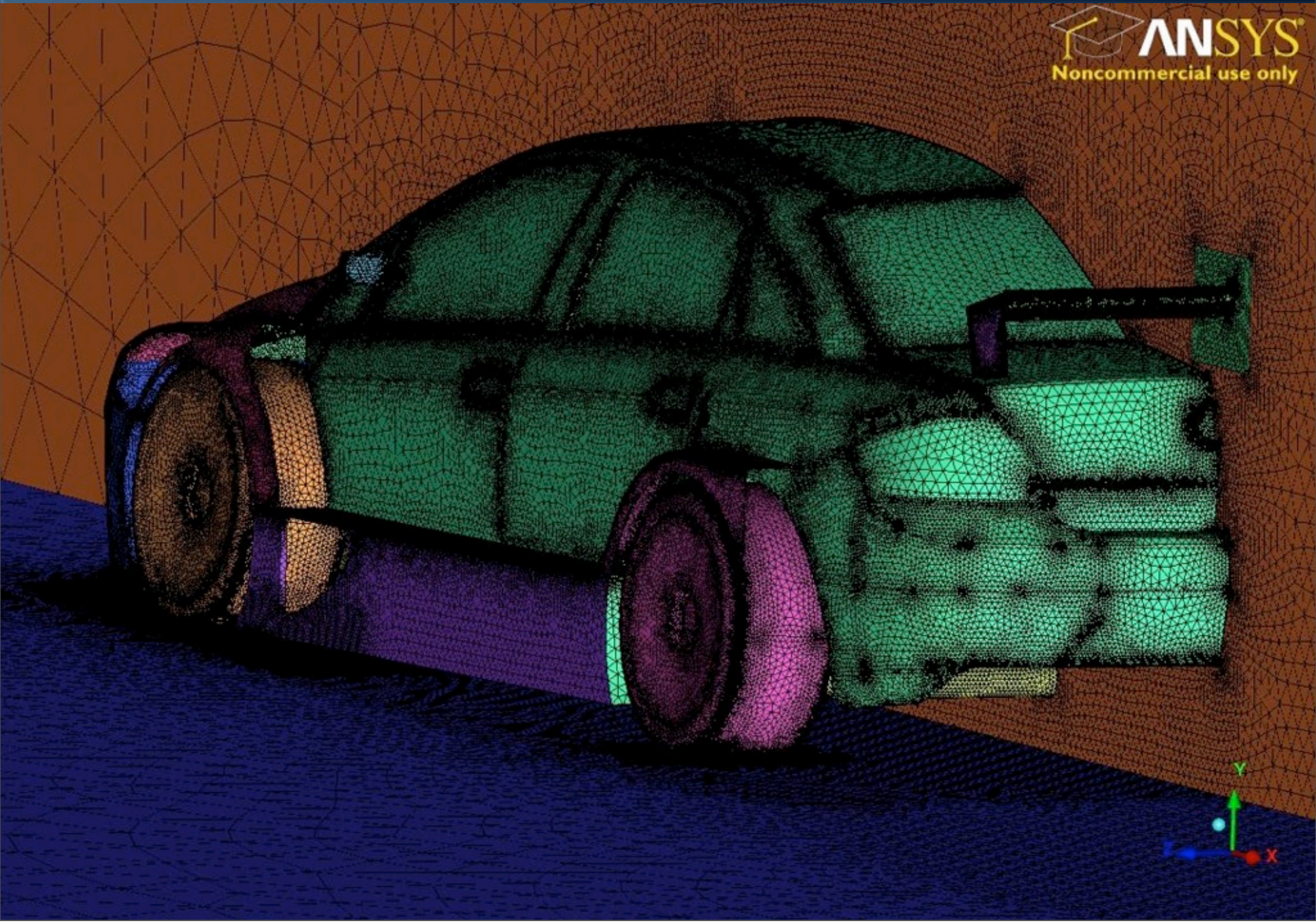
Numerical results - Velocity



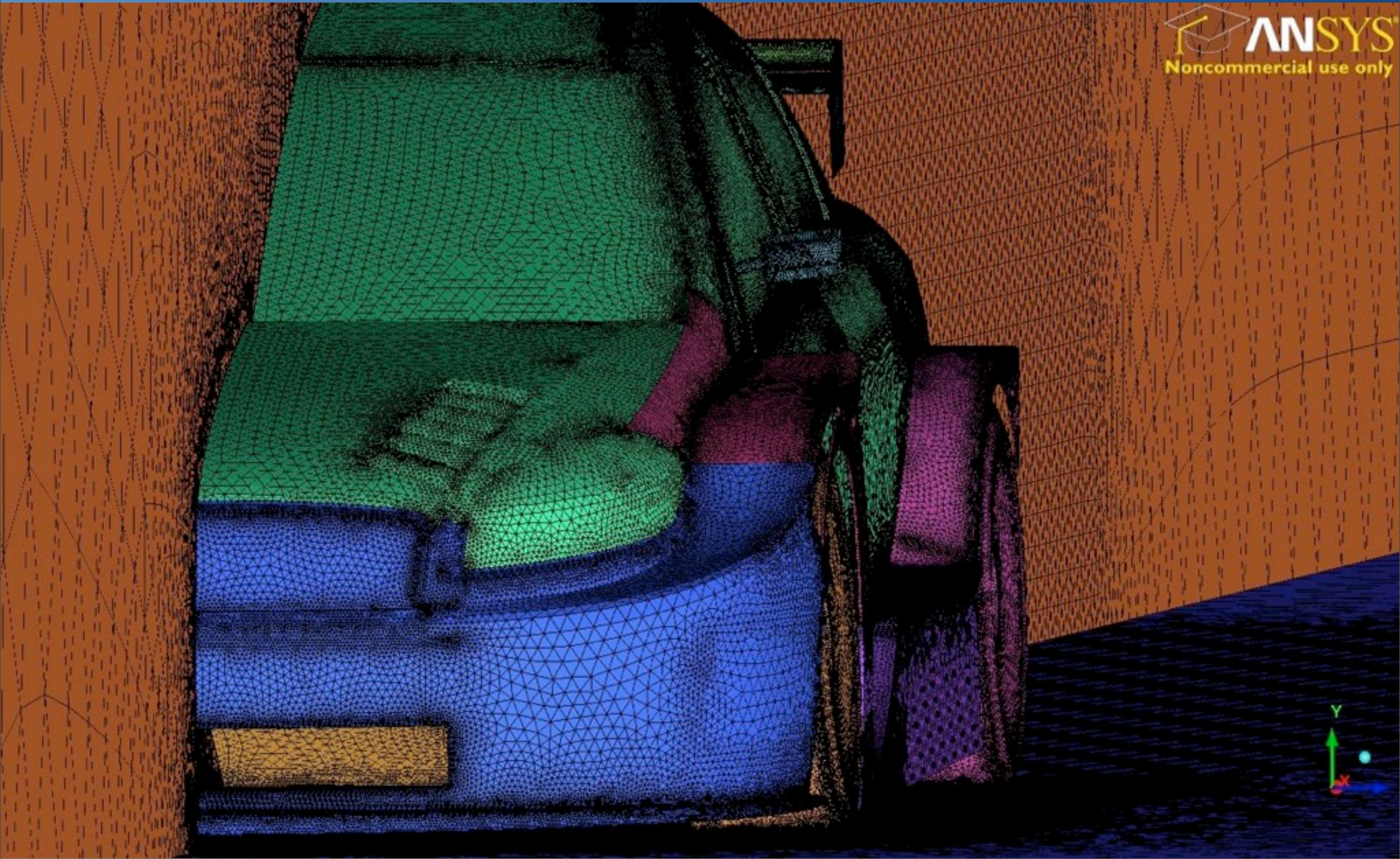
Vehicle Aerodynamics - Mazda 6 - RANS example



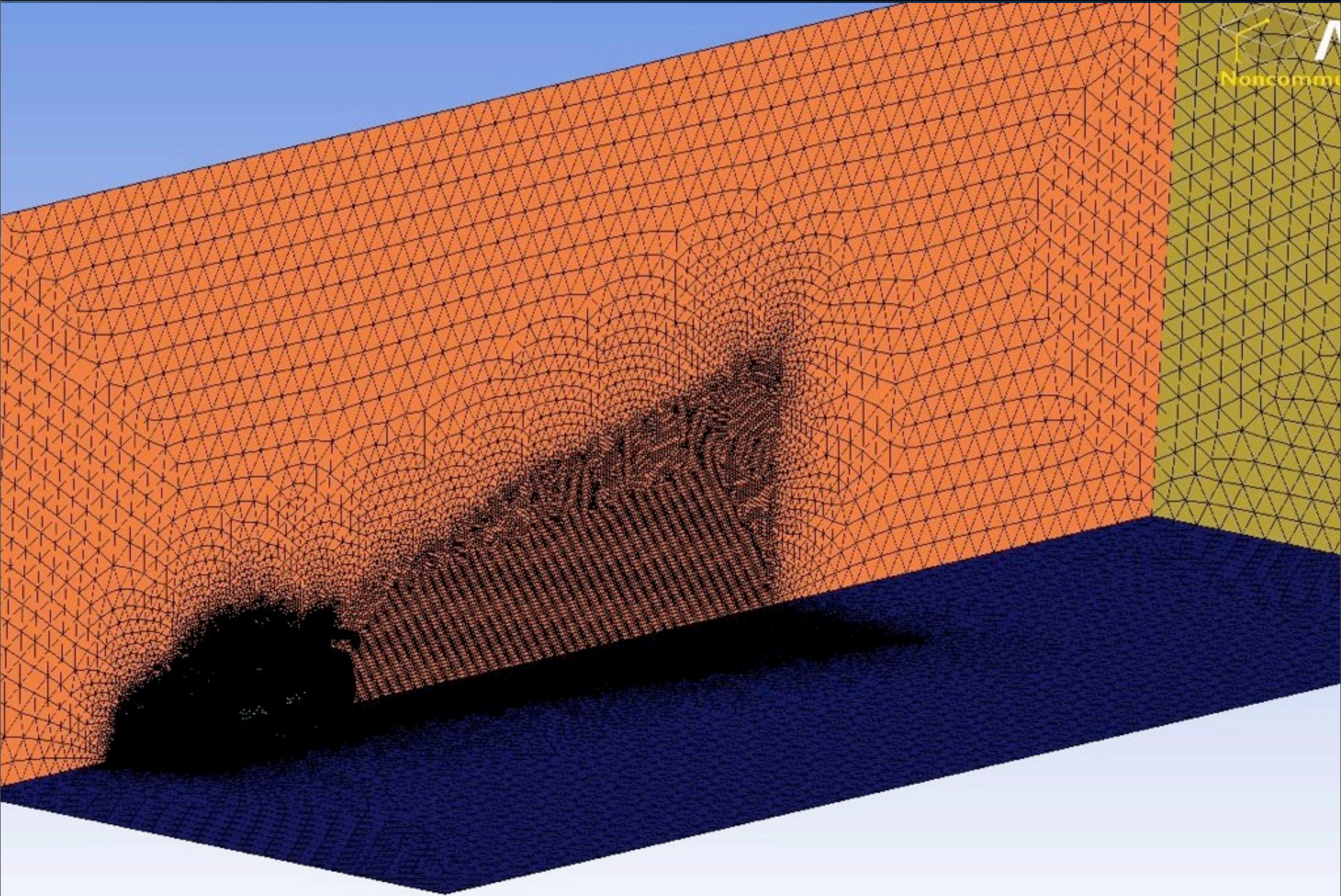
Vehicle Aerodynamics - Race car TC2000 Argentina



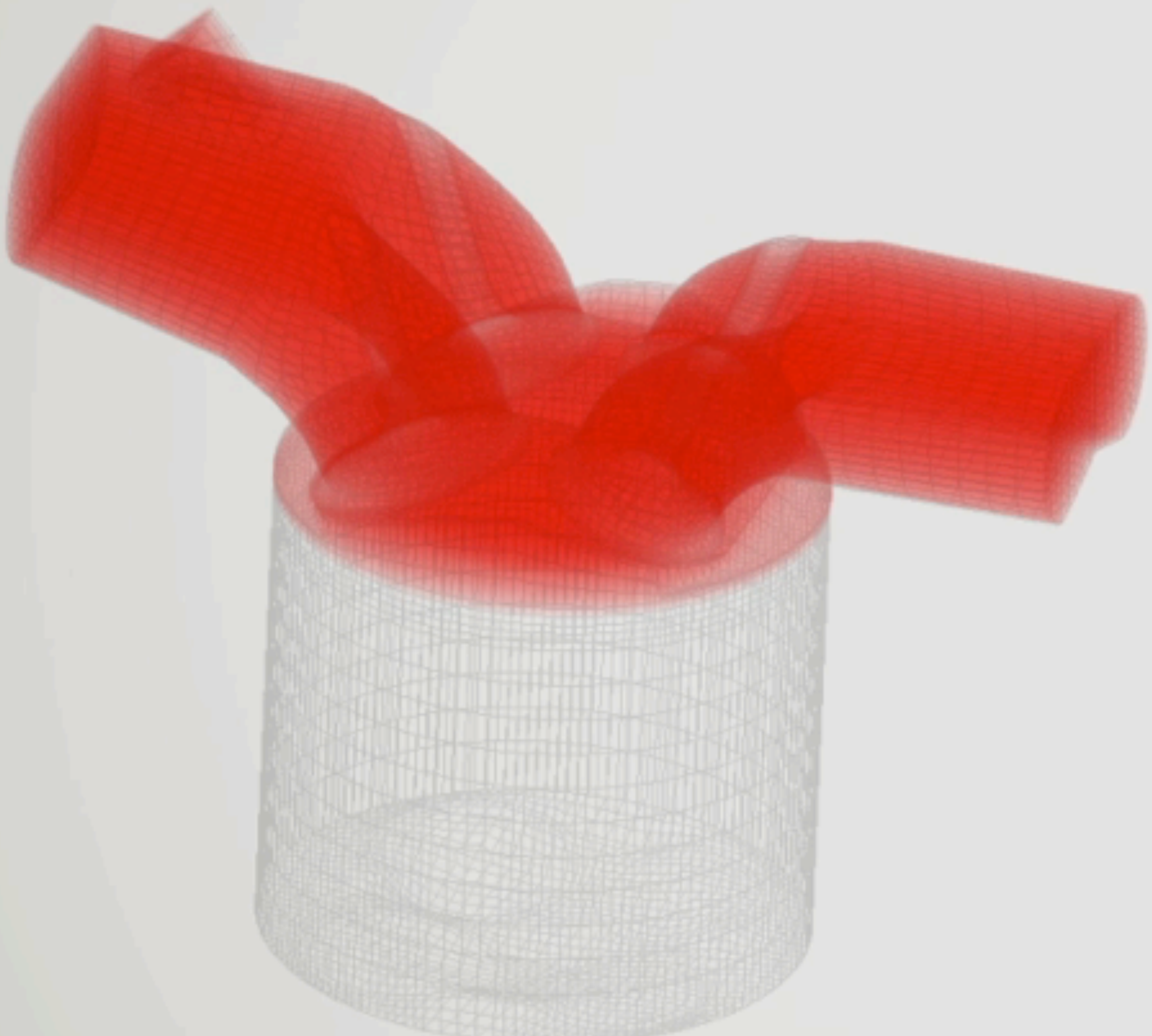
Vehicle Aerodynamics - Race car TC2000 Argentina

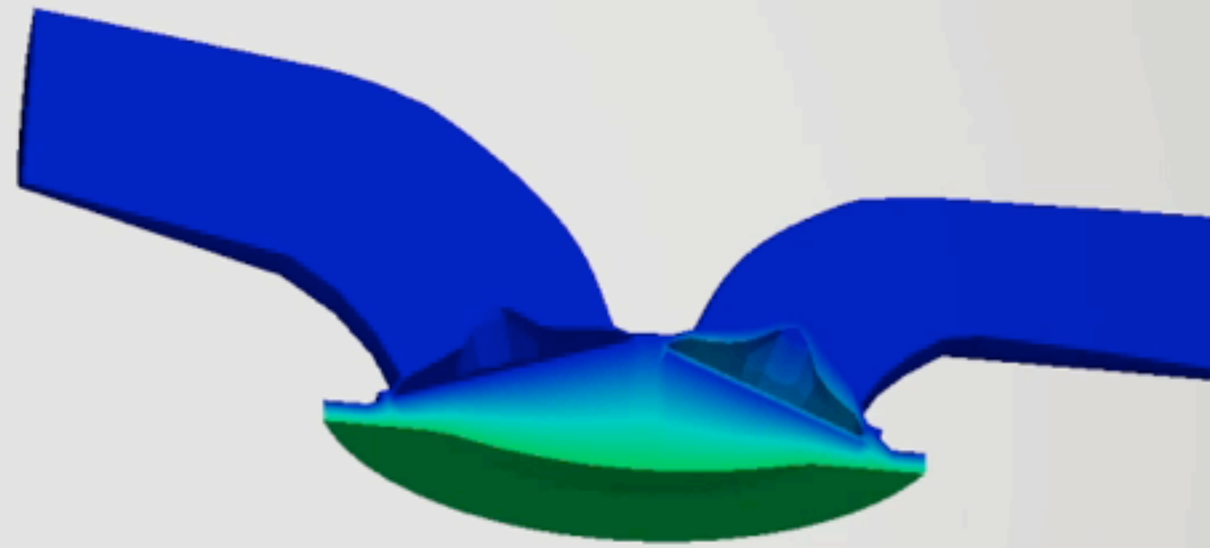


Vehicle Aerodynamics - Race car TC2000 Argentina



4 valves engine - RNG k-epsilon





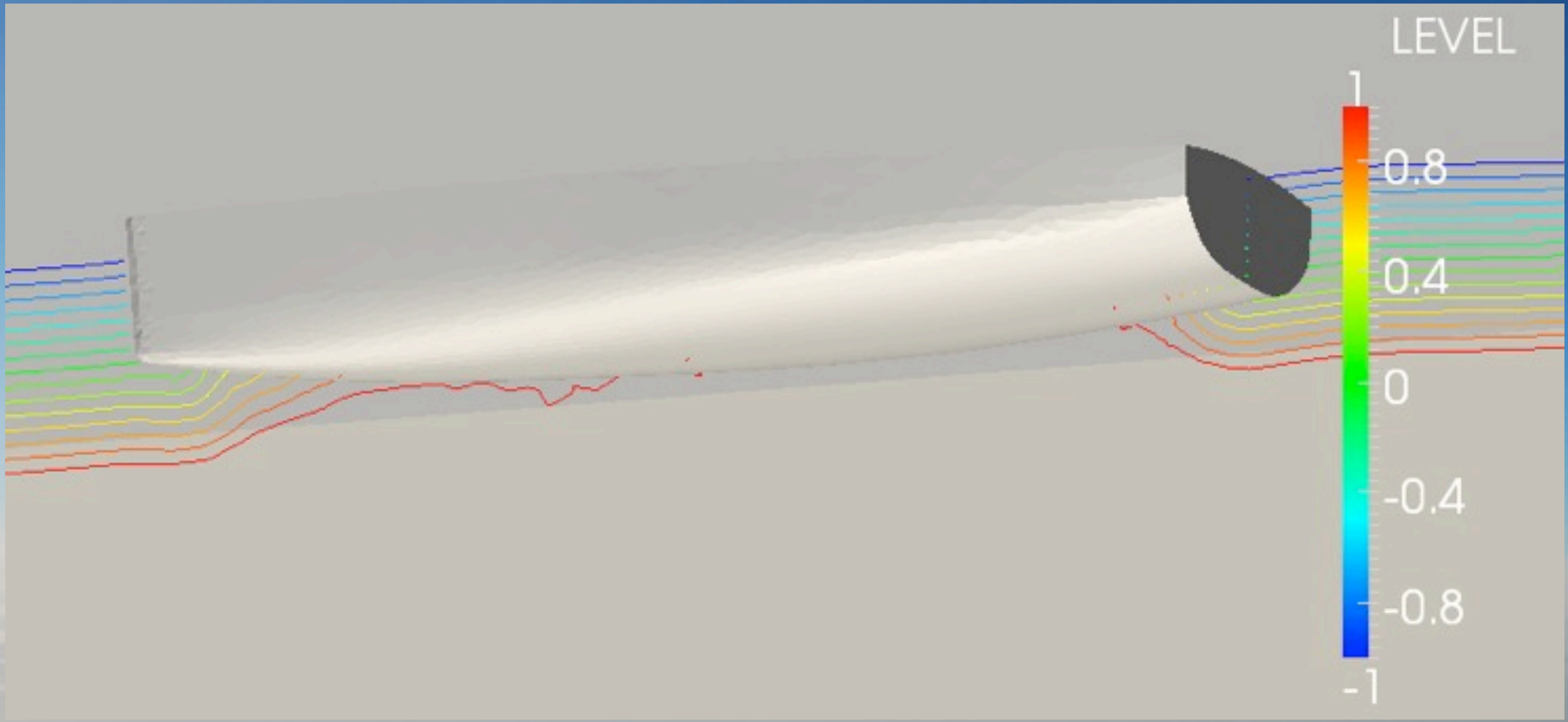
Physical and Numerical Model

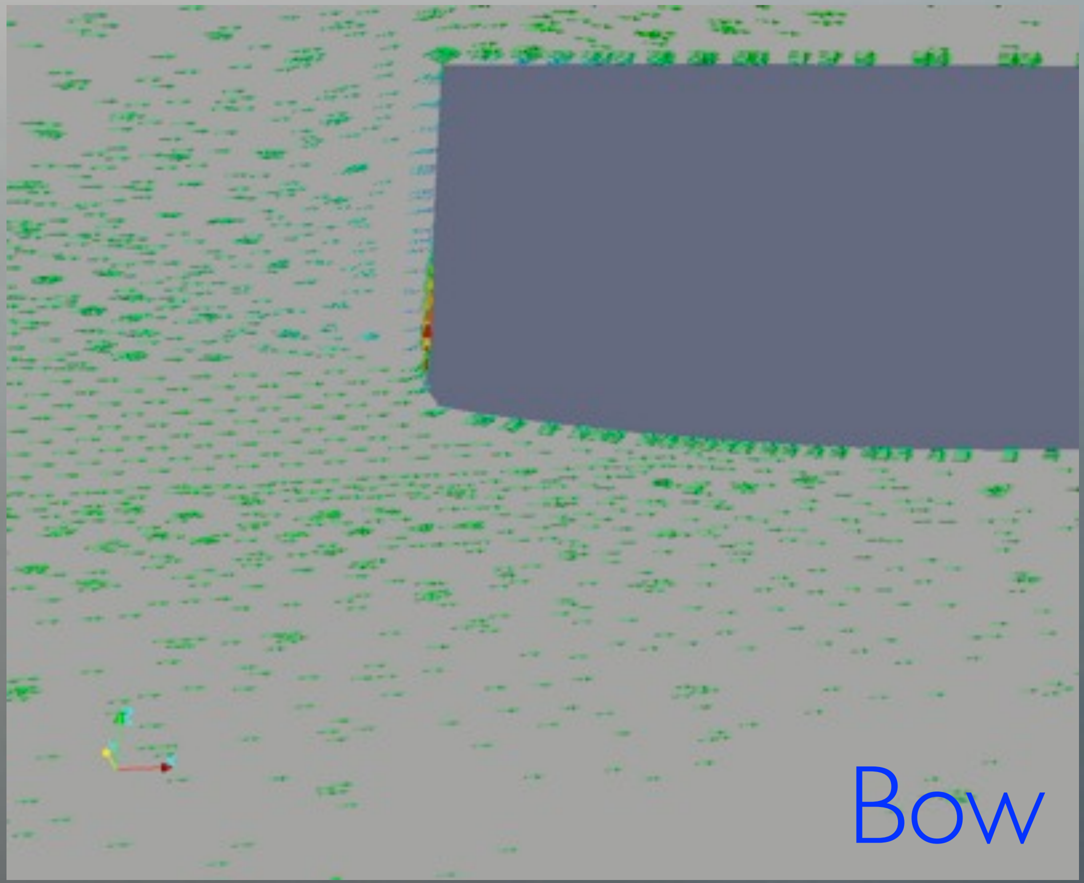
- K-omega SST Turbulence model
- Level set method to capture the position of the interface
- Stabilized Finite element method to solve Navier Stokes, Level Set and turbulence equations.

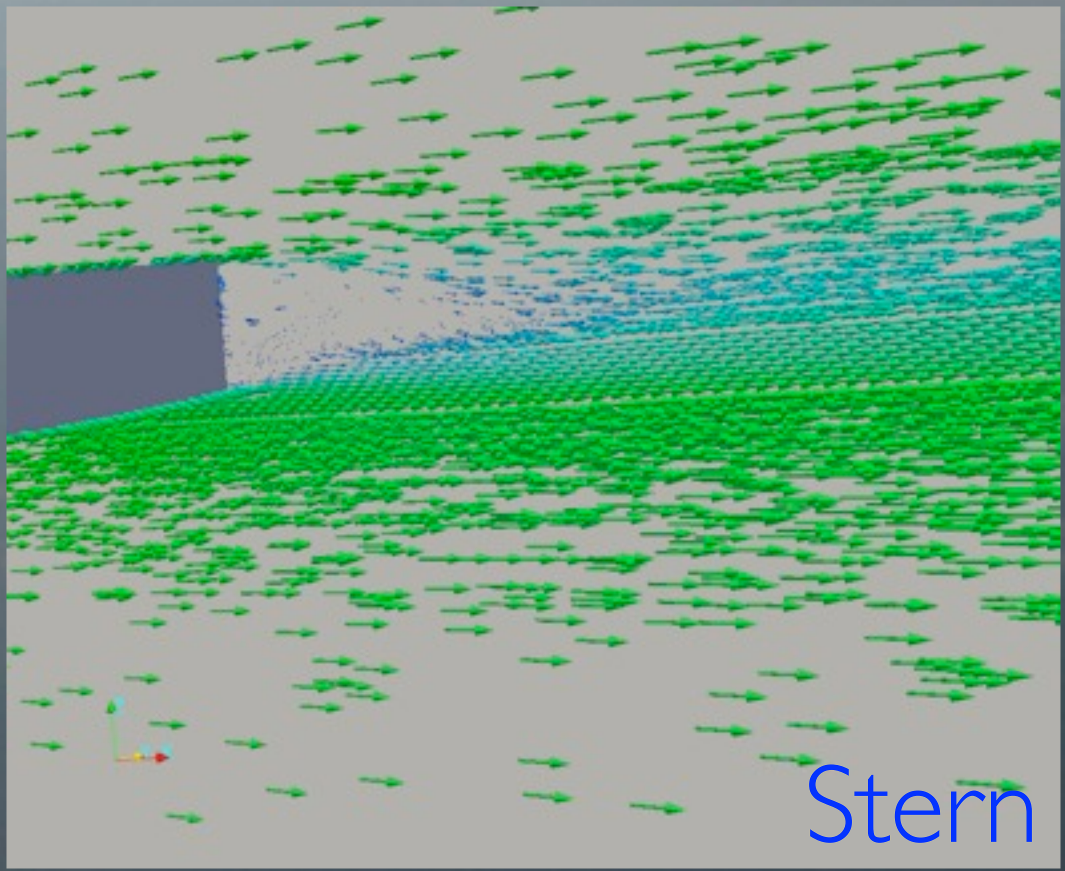
2 Fluid Flow

- Level Set
- Smoothed properties.
- Geometrical redistancing using kdtree.

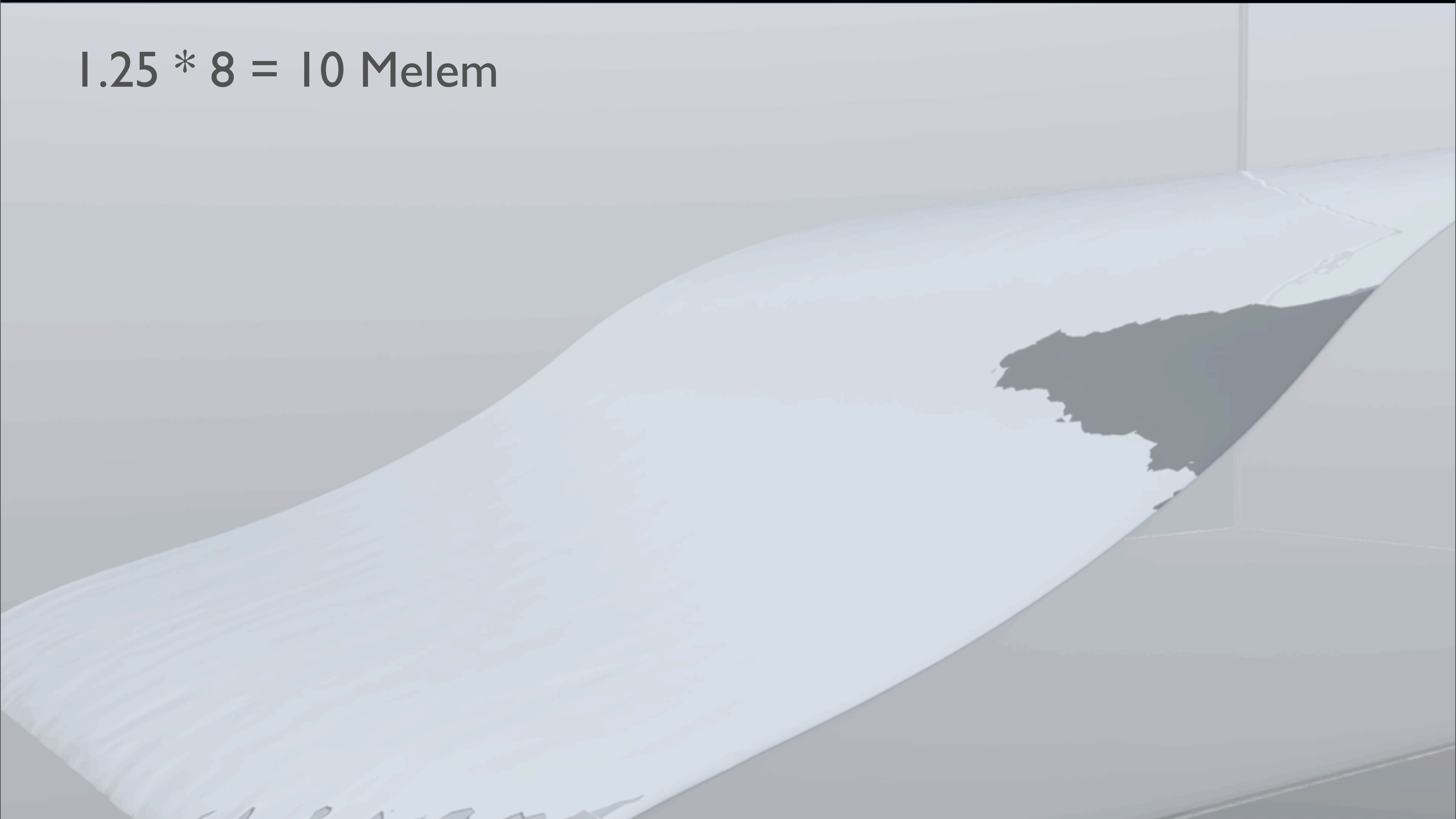
Level Set





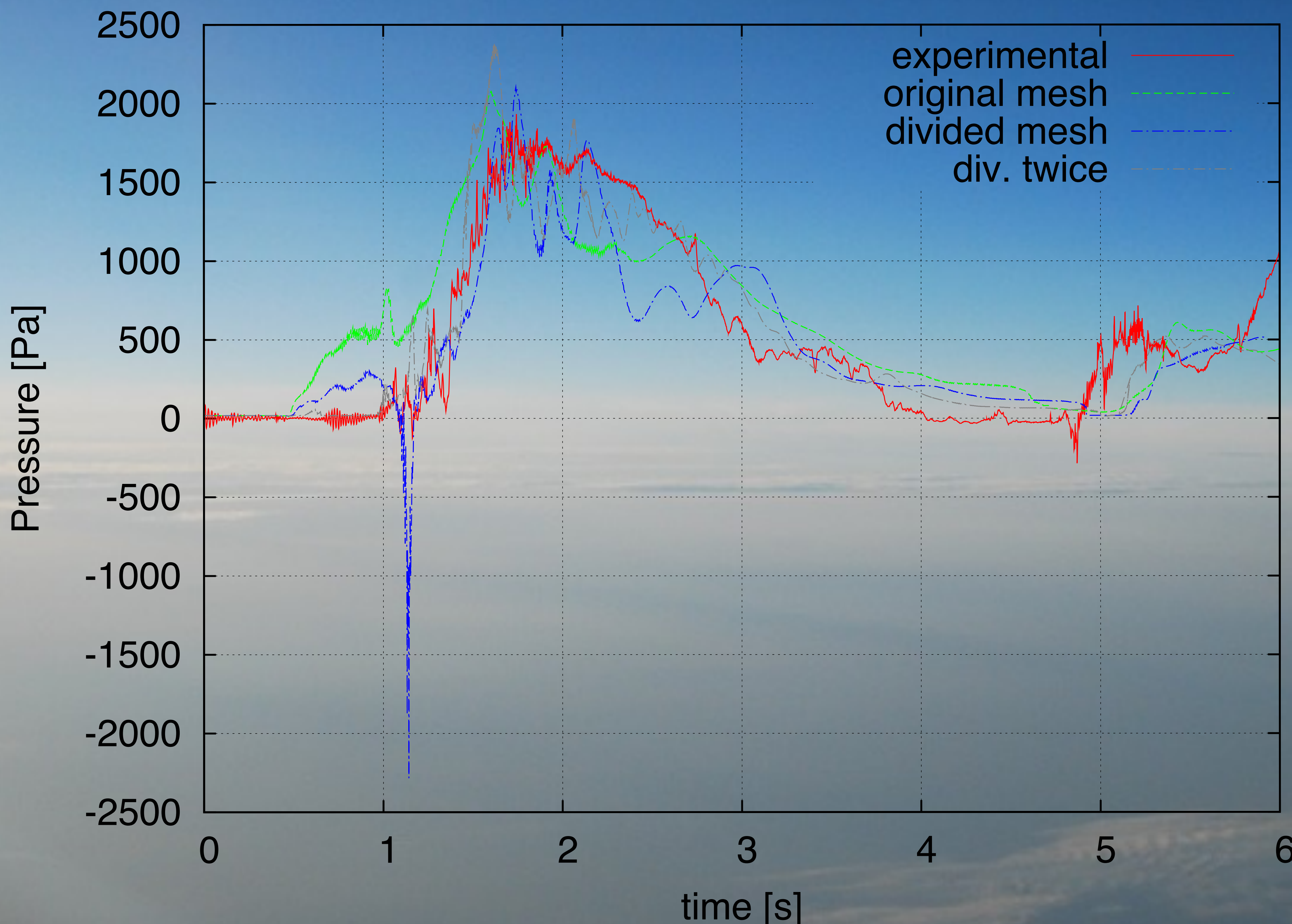


1.25 * 8 = 10 Melem



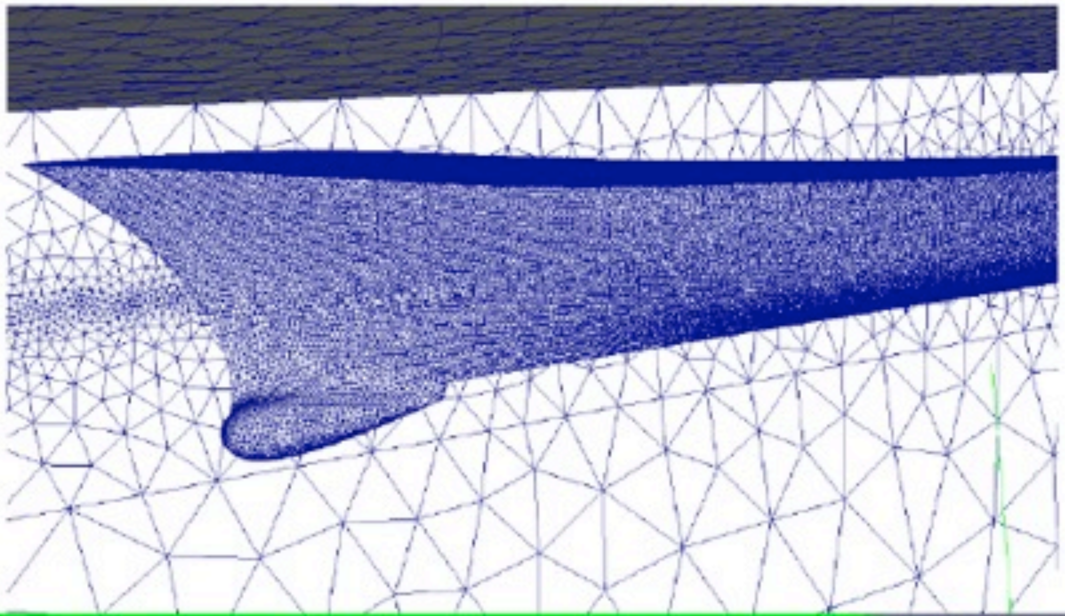
Green Water Dam Break (1div)

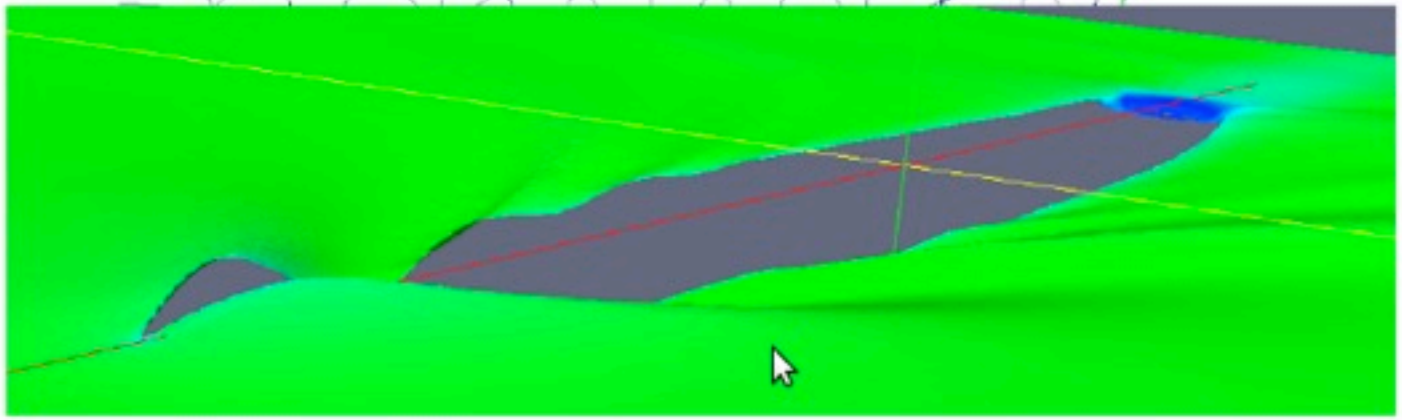
http://wiki.manchester.ac.uk/spheric/index.php



Green Water Dam Break vs. experimental (P6)

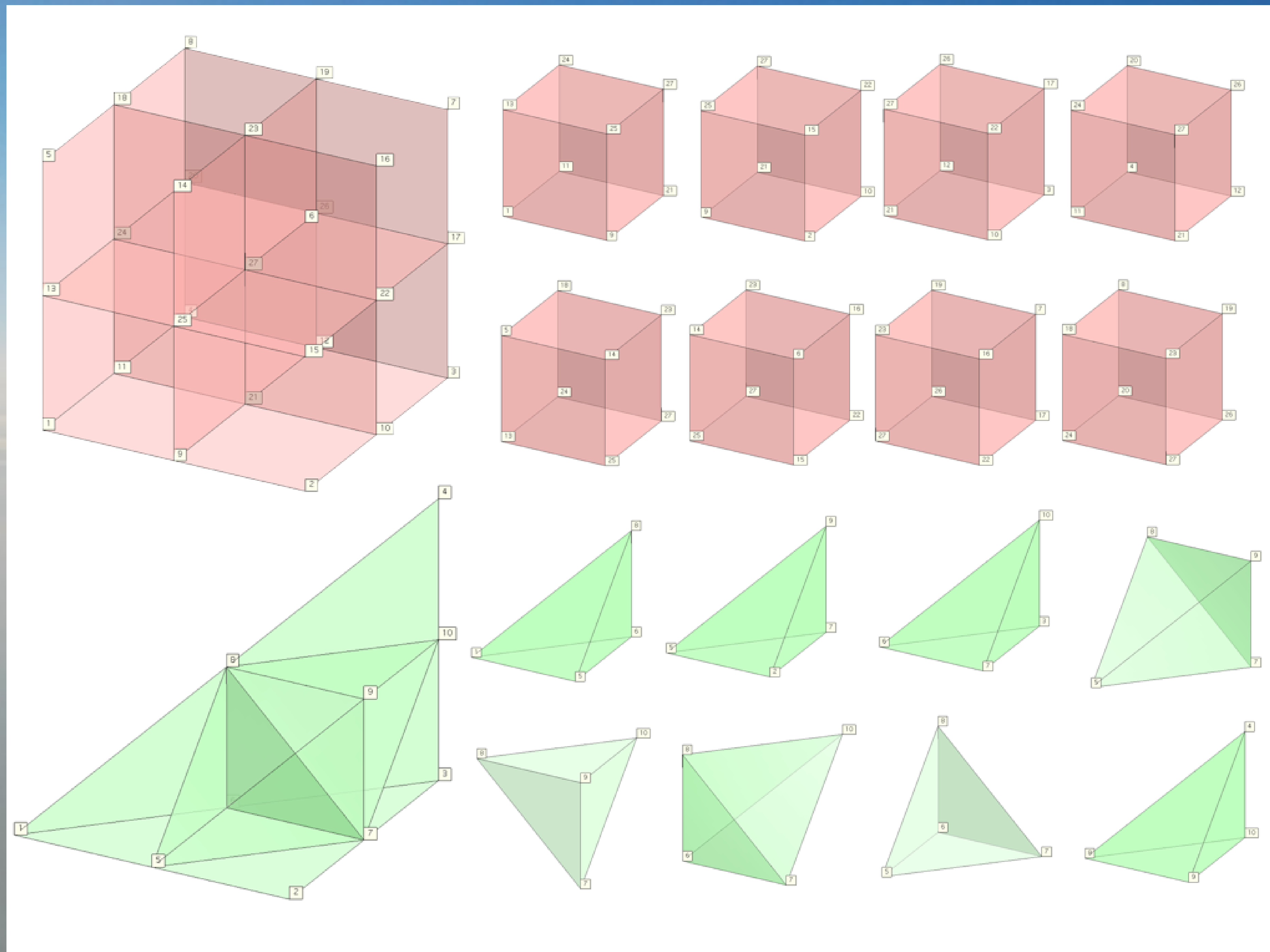
Free surface shape and meshes used





- Mesh A: 8Melem (7.5 tetrahedras + 0.5 prisms).
- Mesh B: 5Melem (4.6 tetrahedras + 0.4 prisms). Half domain - Symmetry.
- Mesh C: 40Melem. Previous mesh divided into elements with half the size.

ALYA – Automatic DIVISOR



Guillaume Houzeaux, Raúl de la Cruz, Herbert Owen, and Mariano Vázquez. Parallel uniform mesh multiplication applied to a Navier- Stokes solver. *Computers and Fluids*

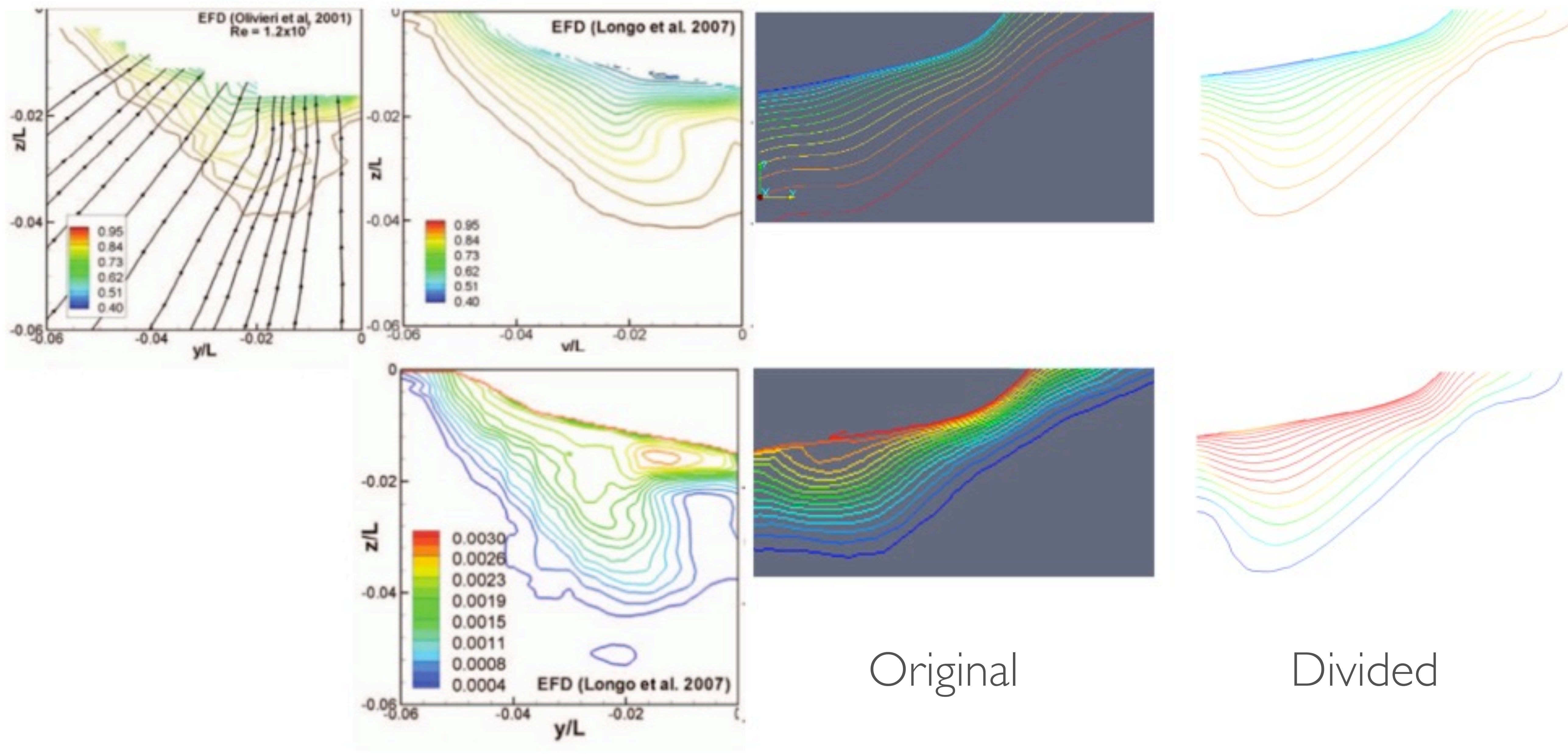
Benchmarking

•David Taylor Model Basin model - DTMB 5512.

•1:46.6 model scale of a modern surface combatant

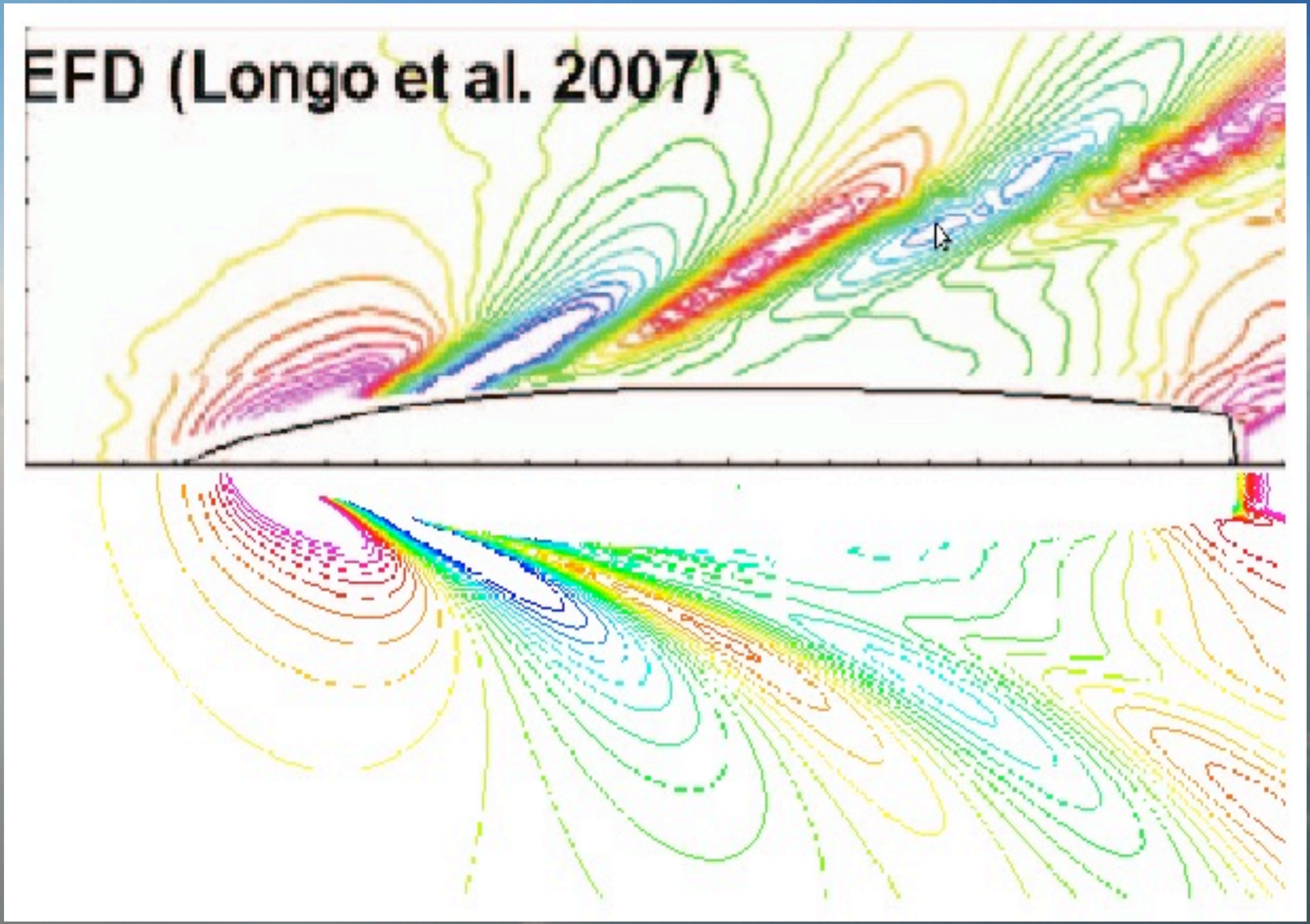
•It has been tested in the towing tanks at DTMB, IIHR(Iowa) and INSEAN (Italy)

Velocity and turbulent kinetic energy comparison

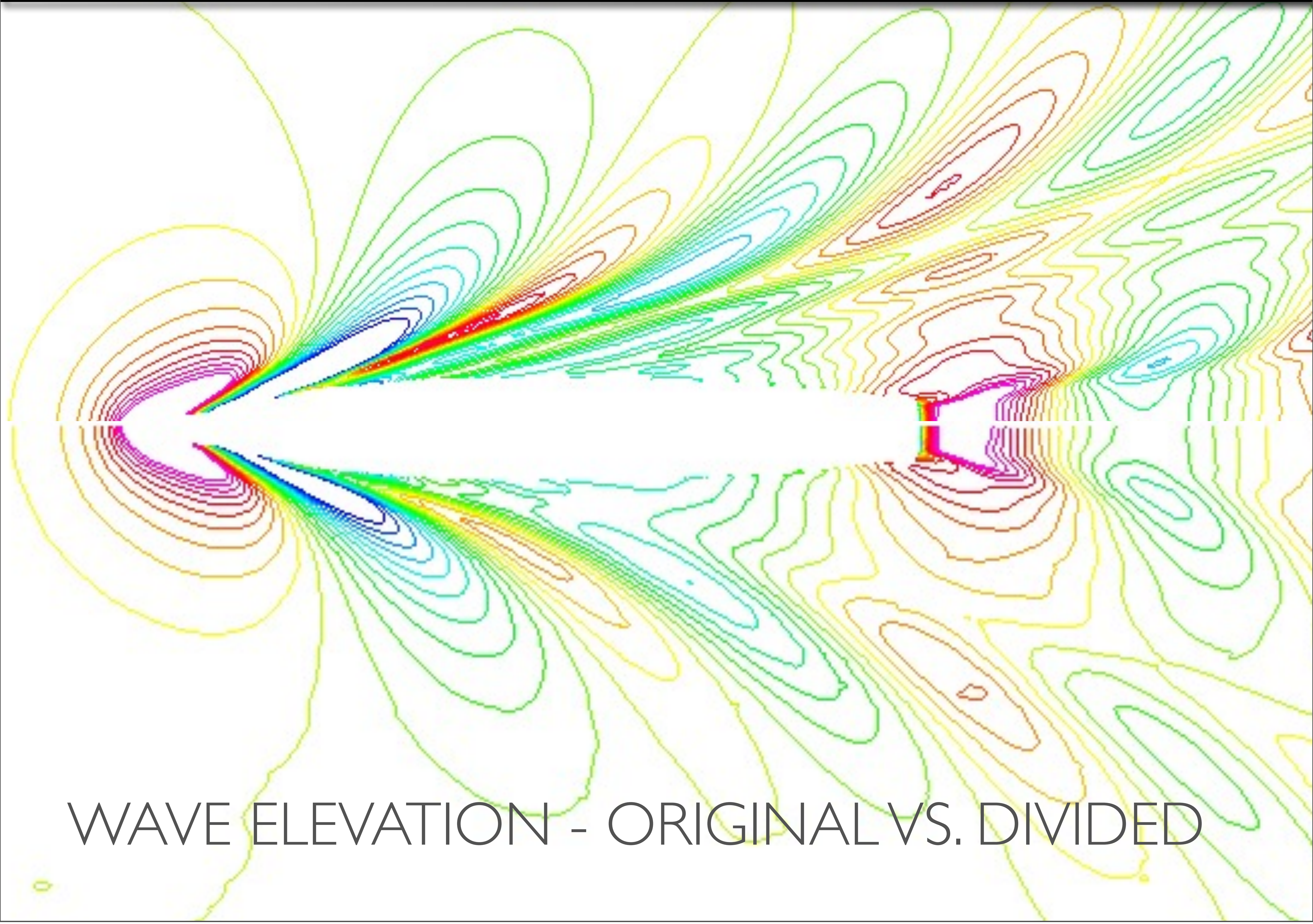


Results at x/L=0.95. from: J. Longo, J. Shao, M. Irvine, and F. Stern. Phase-averaged piv for the nominal wake of a surface ship in regular head waves. Journal of Fluids Engineering, 129:524–540, 2007.

Wave elevation - comparison with experimental results



Non dimensional wave elevation profiles for the 5512 hull. Top: experimental data, bottom: numerical result



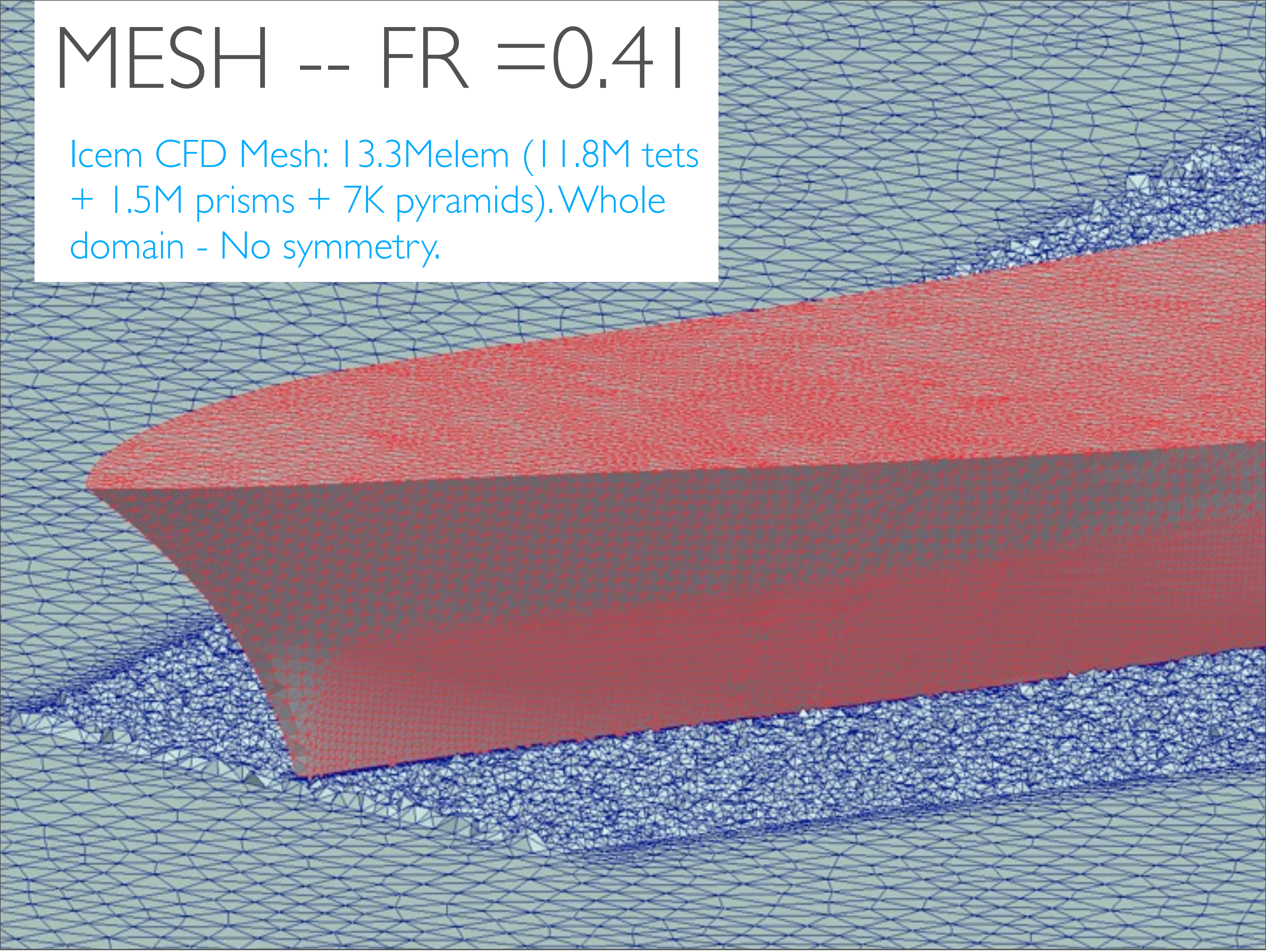
WAVE ELEVATION - ORIGINAL VS. DIVIDED

Forces on the hull Fr = 0.28

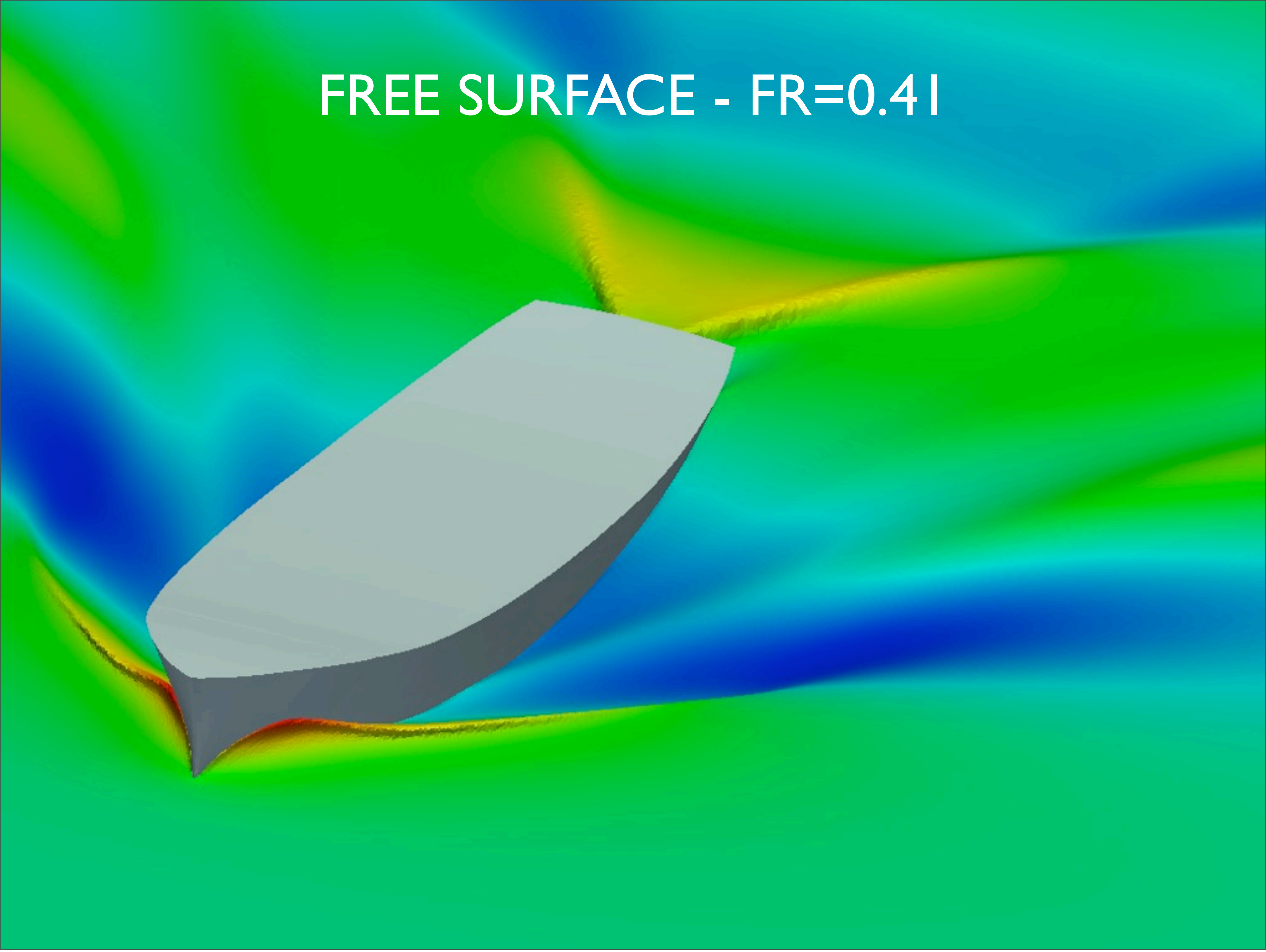
	Experimental	Original Mesh	Divided Mesh
Fx[N] (total)	7.432	7.41	7.30
Fvx[N] (viscous)	5.52 (from correlation)	5.54	5.54
Fpx[N](pressure)		1.87	1.76

MESH -- FR =0.41

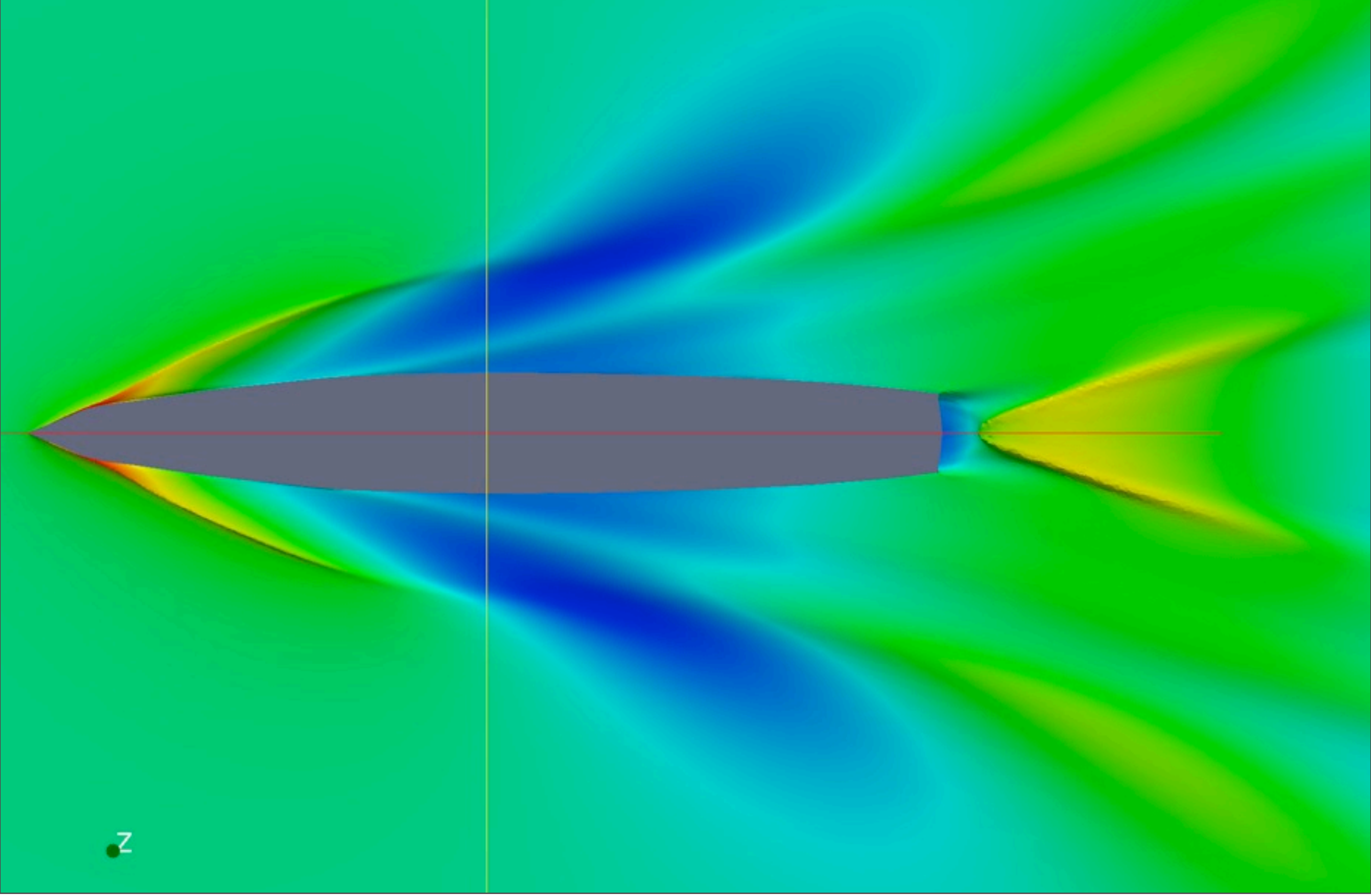
Icem CFD Mesh: 13.3Melem (11.8M tets + 1.5M prisms + 7K pyramids). Whole domain - No symmetry.



FREE SURFACE - FR=0.41



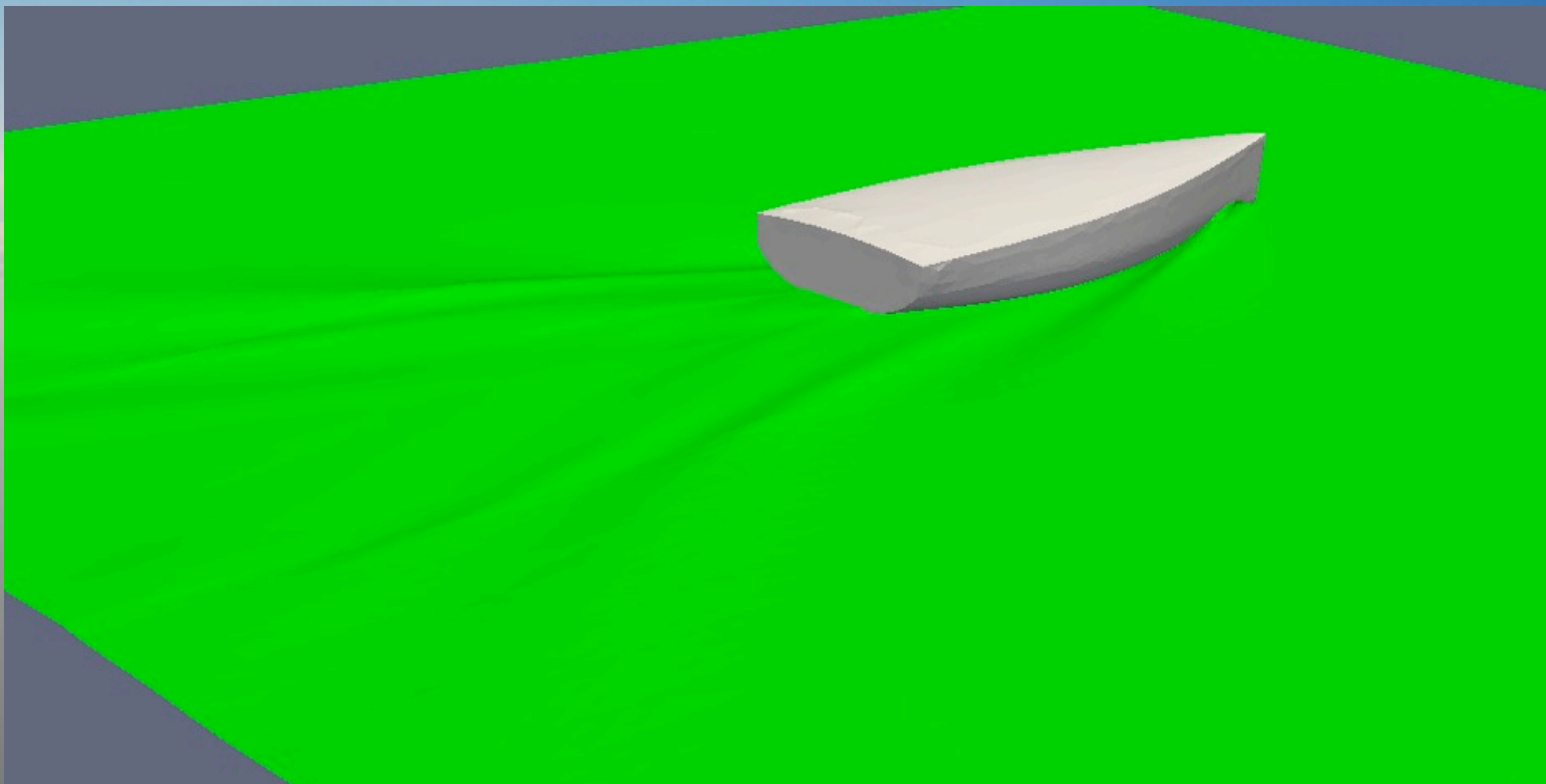
FREE SURFACE - FR=0.41



Forces on the hull Fr = 0.41

	Experimental	Numerical
Fx[N] (total)	23.52	24.54
Fvx[N] (viscous)	10.9 (from correlation)	11.64
Fpx[N](pressure)		12.9

Application to a race boat hull



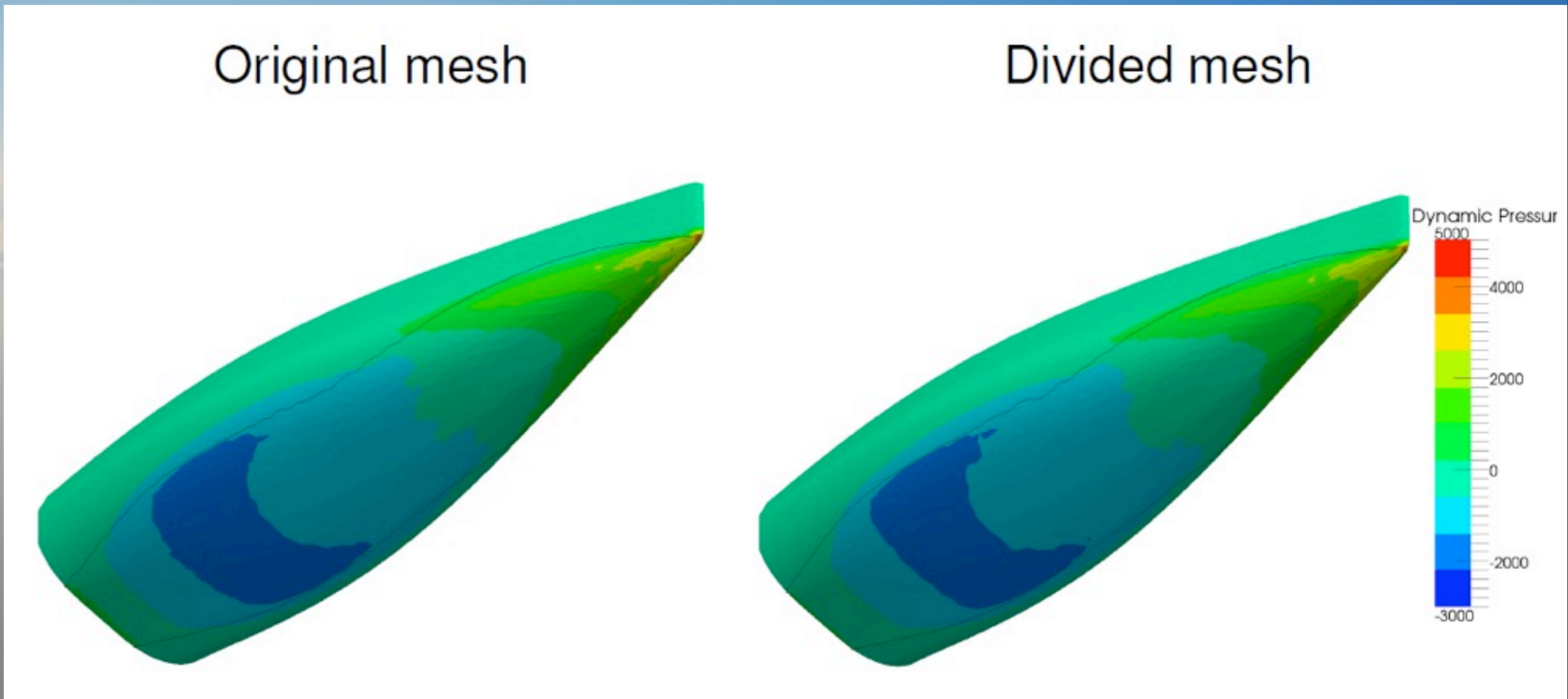
Application to a race boat hull

Length, L	$10.85m$
Velocity, U	$5.144m/s$
$Re = \frac{UL\rho}{\mu}$	$4.7 * 10^7$
$Fr = \frac{U}{\sqrt{gL}}$	0.4985

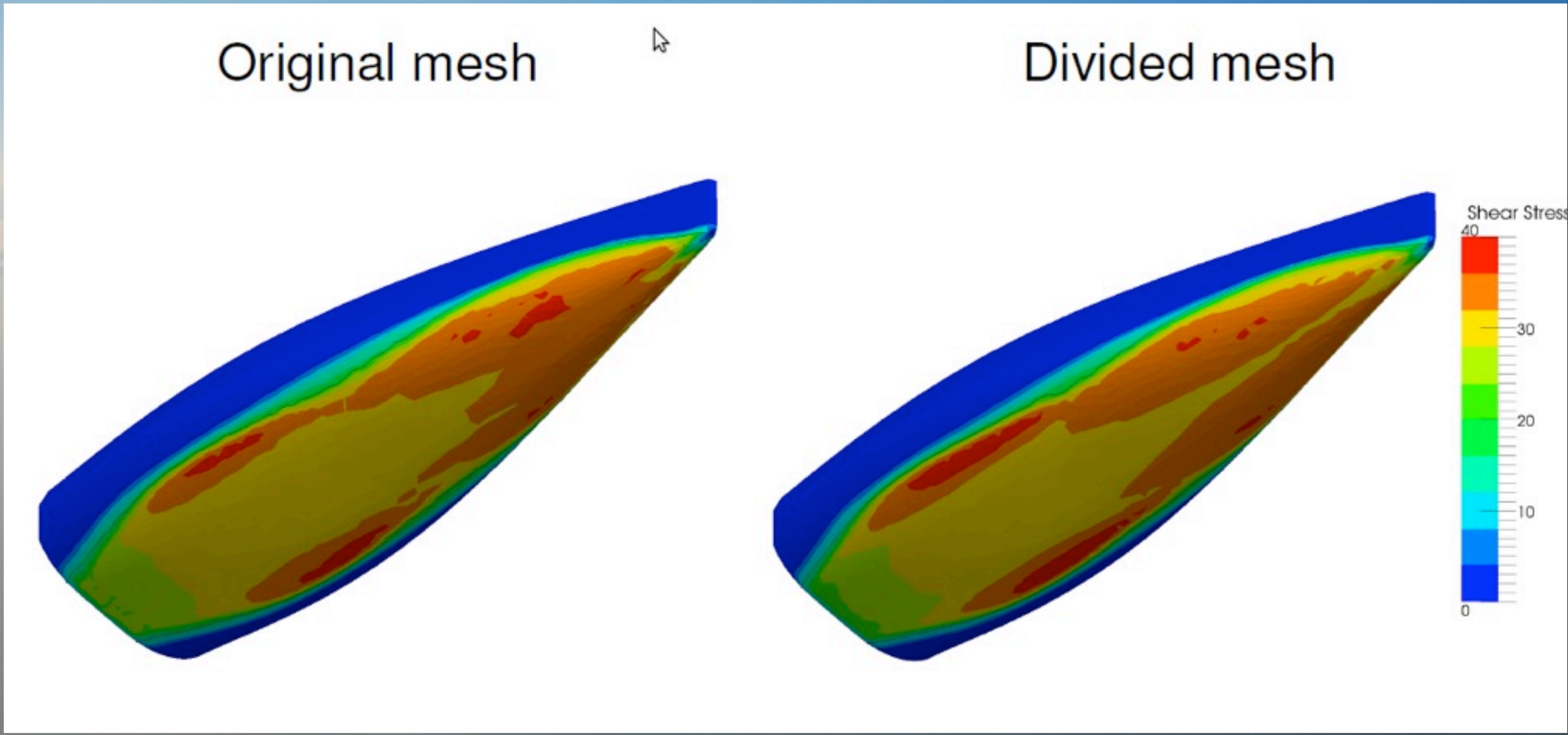
- Original mesh: 2.12 Melem (Tetras + Prisms), 0.36 Gnodes
- Divided mesh: Each element is divided so that the new element has half the size. In 3D 8 times more nodes and elements.

	Original mesh	Divided mesh
$F_x[N](total)$	2132.0	2161.6
$F_{Vx}[N]$ (viscous)	744.0	748.6
$F_{Px}[N]$(pressure)	1388.0	1413.0

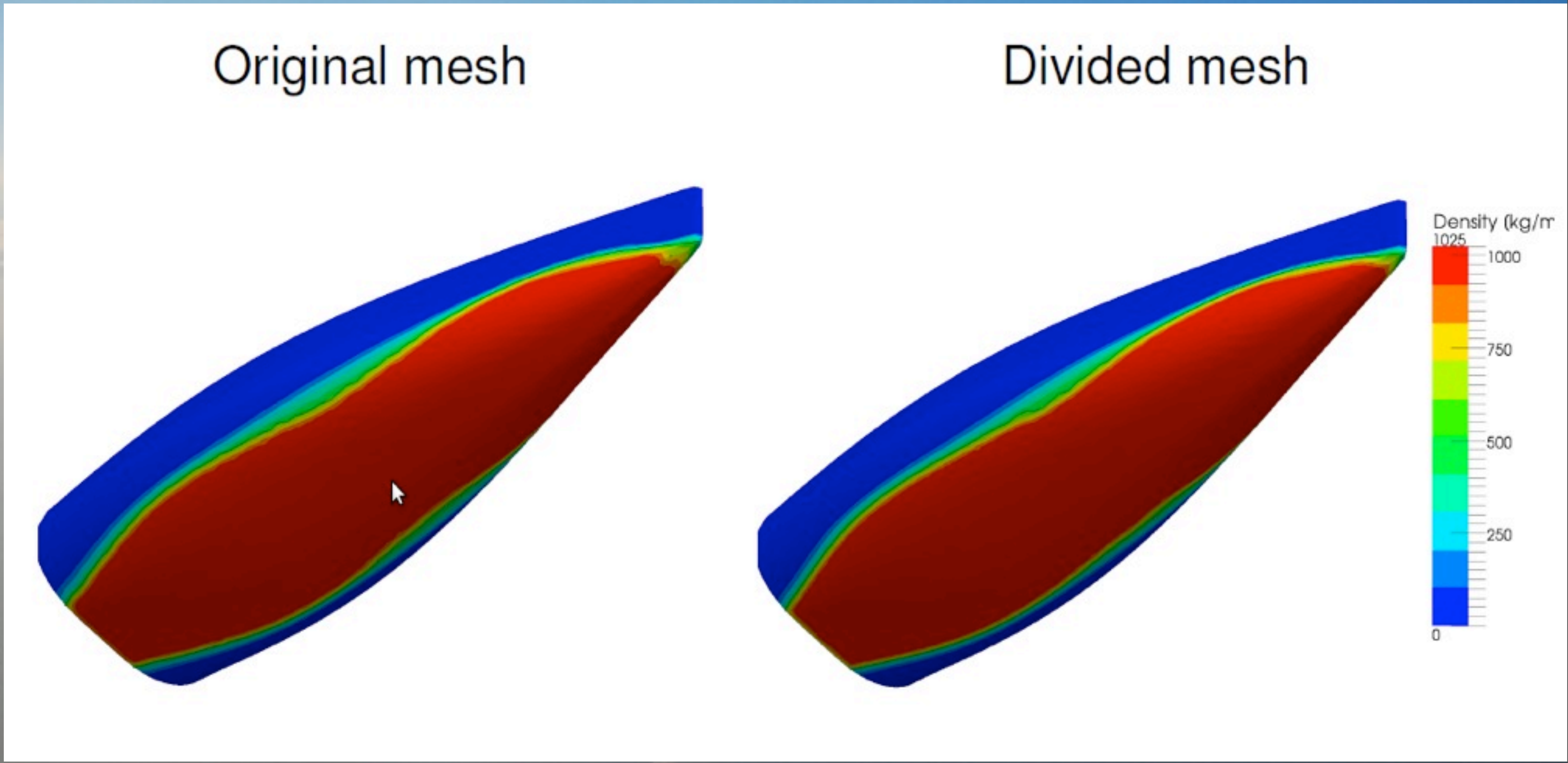
Pressure


Original mesh
Divided mesh
Dynamic Pressur
5000
4000
2000
0
-2000
-3000

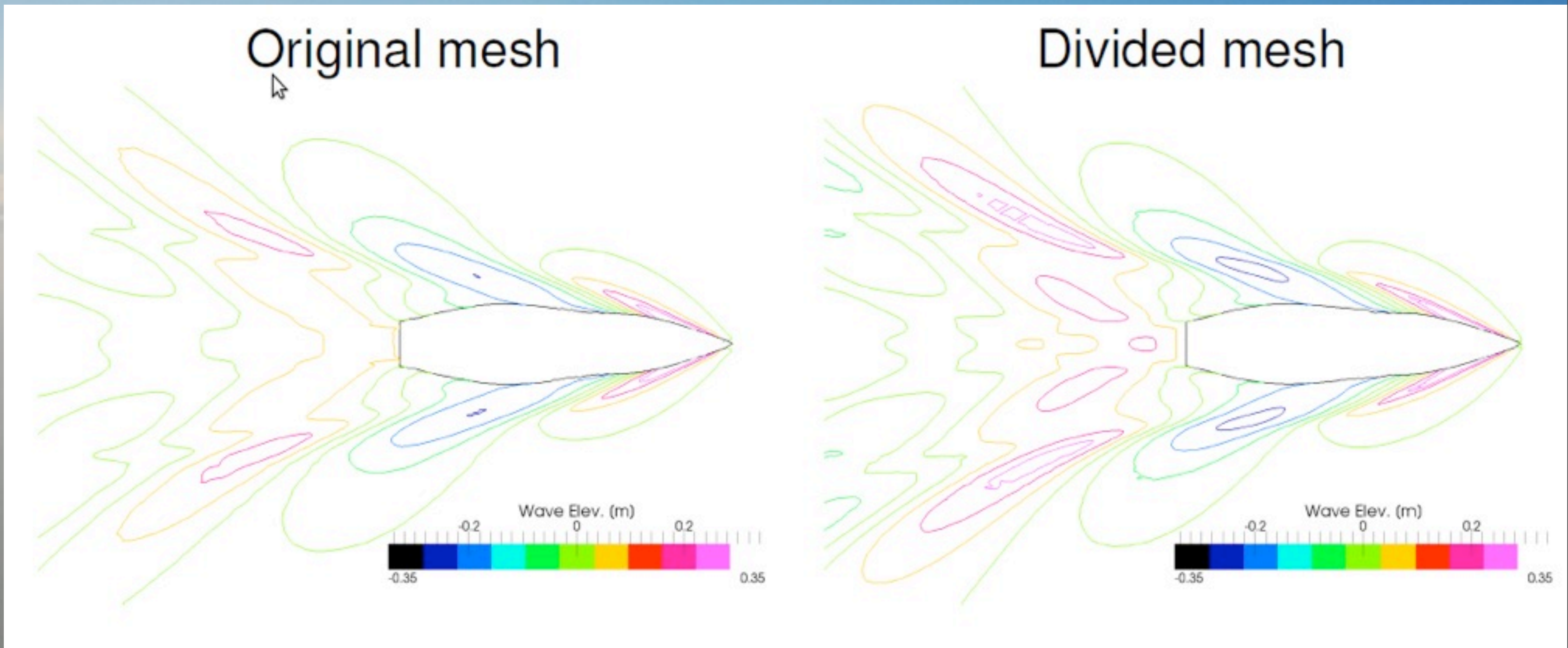
Tangential Stress



Density


Original mesh
Divided mesh
Density (kg/m
1025
1000
750
500
250
0

Wave elevation



Linear and angular RB dynamics

$$m\ddot{\mathbf{X}} = \mathbf{F},$$

$$\mathbf{RIR}^{-1}\dot{\mathbf{\Omega}} + \mathbf{\Omega} \times \mathbf{RIR}^{-1}\mathbf{\Omega} = \mathbf{M}_G.$$

Time integration scheme

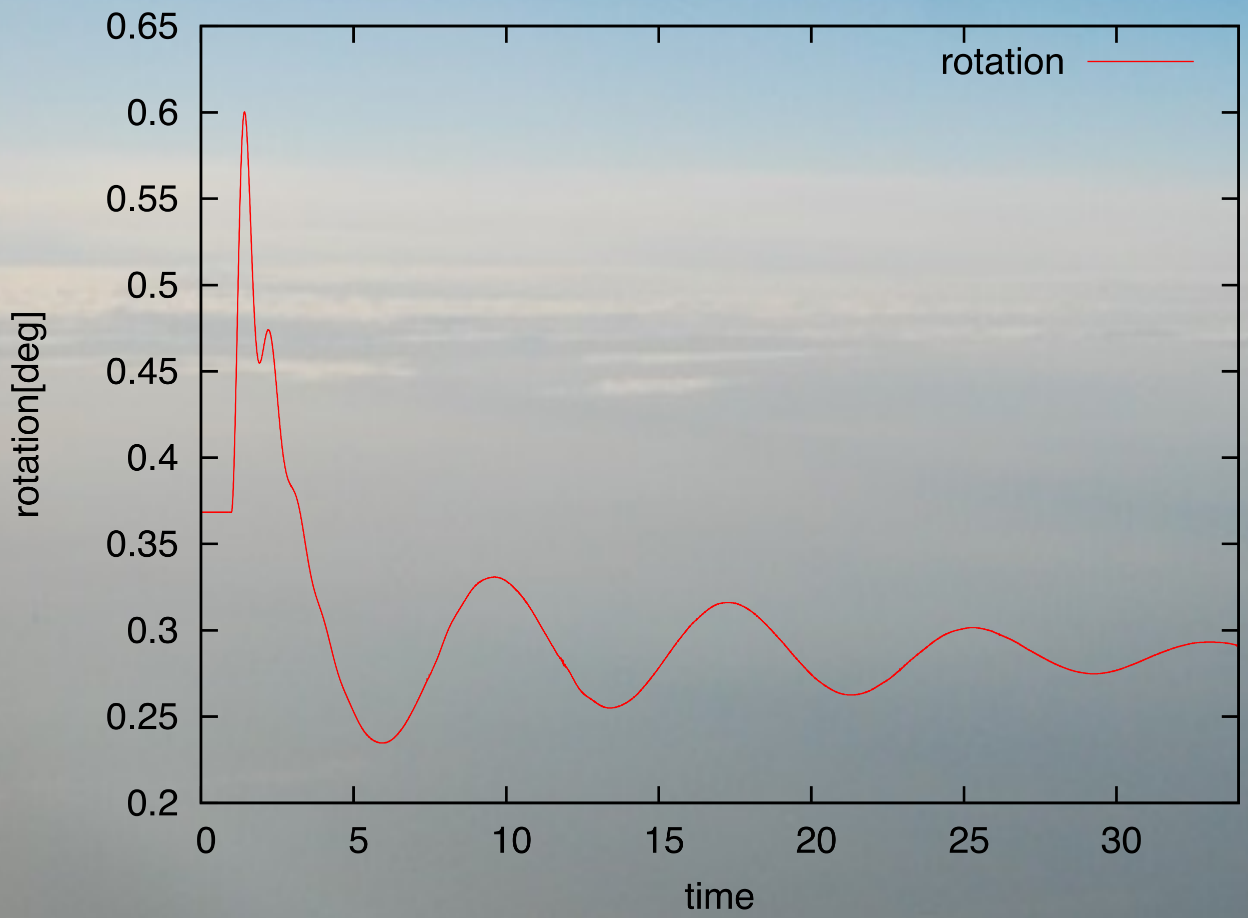
$$\dot{\mathbf{X}}^{n+1} = \dot{\mathbf{X}}^{n} + \delta t \left(1.5\ddot{\mathbf{X}}^{n} - 0.5\ddot{\mathbf{X}}^{n-1}\right)$$

$$\mathbf{X}^{n+1} = \mathbf{X}^{n} + \delta t\dot{\mathbf{X}}^{n+1}.$$

Arbitrary Lagrangian Eulerian formulation

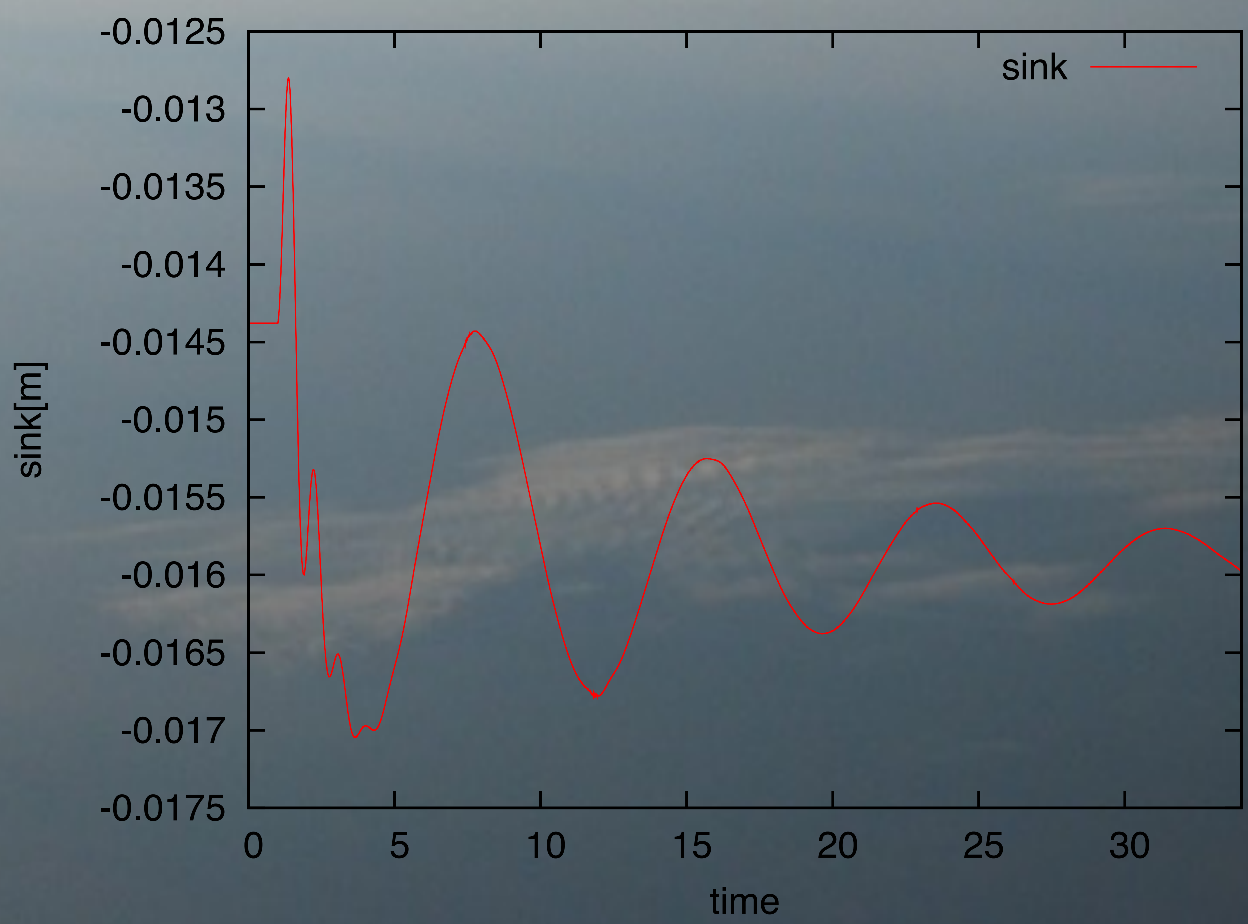
- Convective term in NSI, TUR & LS modified accordingly.
- Rigid body nodes displaced according to RB motion.
- Weighted Laplacian for interior nodes.

Sink and Trim - Fr=0.41

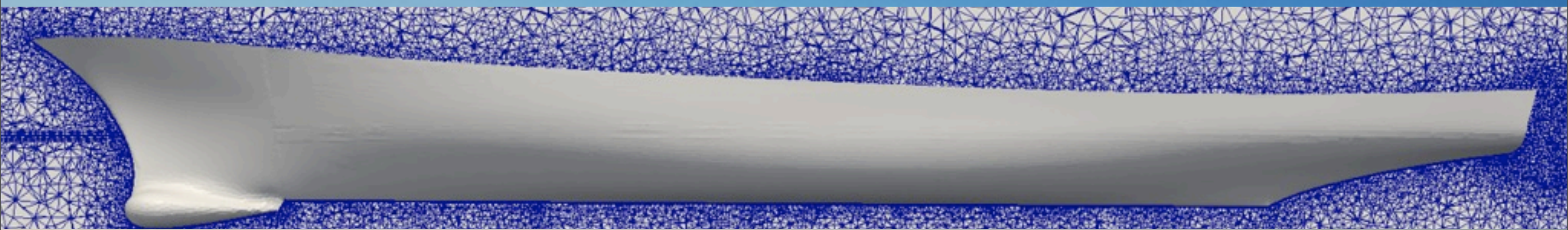


Trim (exp 0.3684)

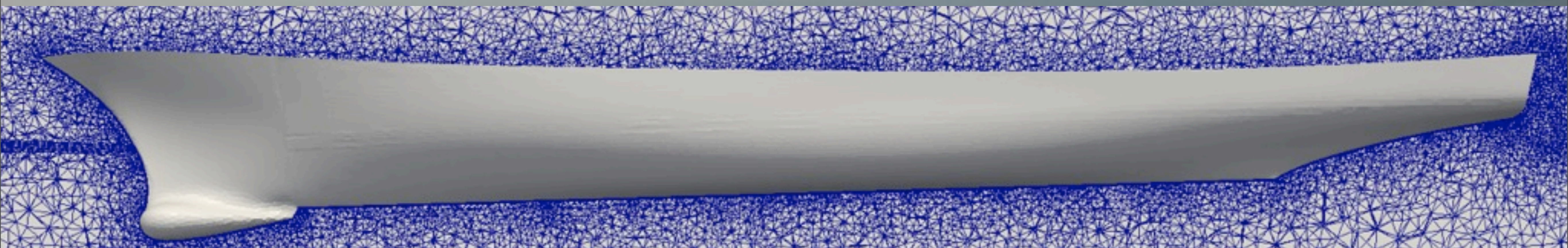
Sink (exp 0.0143)



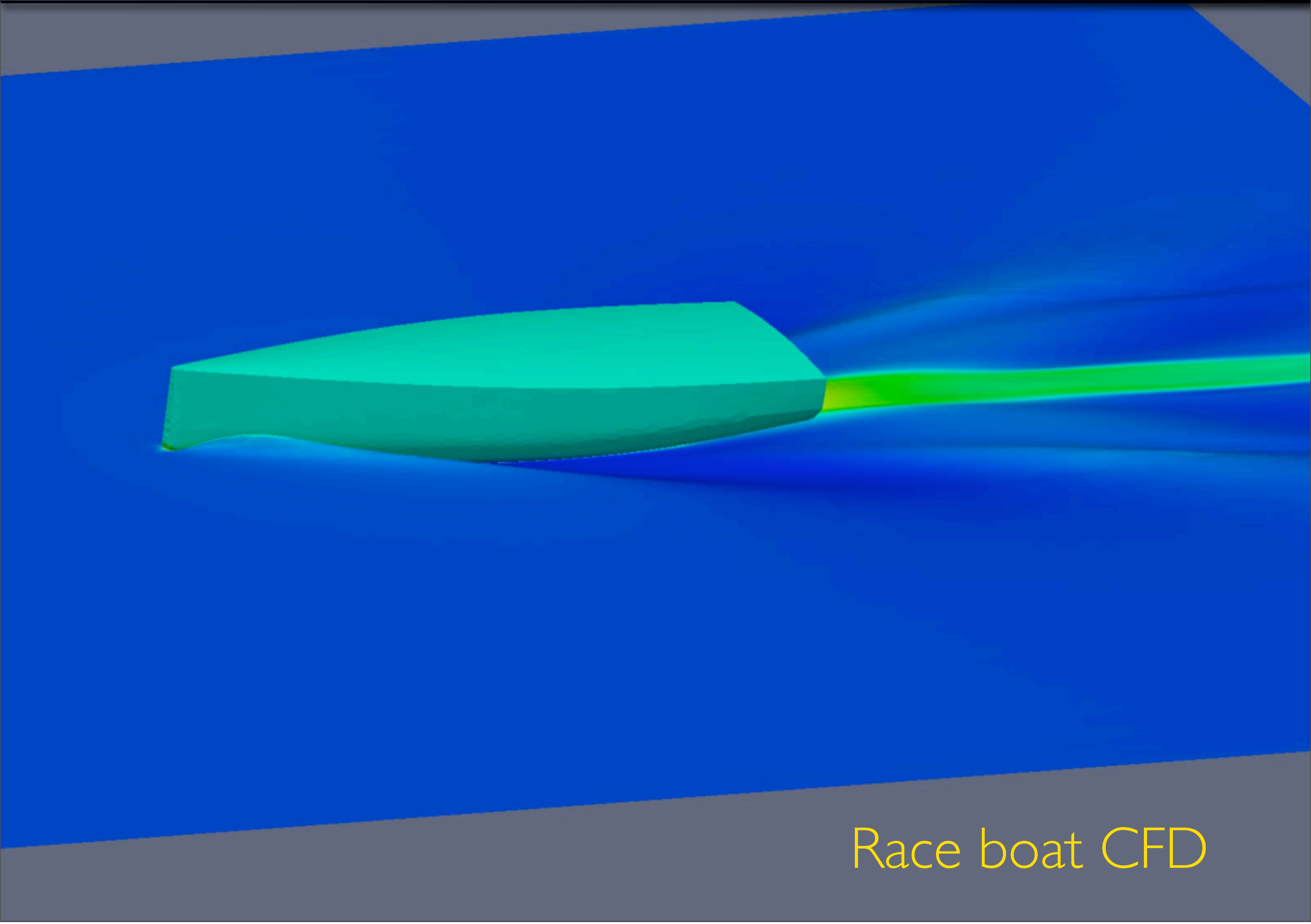
Rigid body motion using ALE - Fr =0.28



Initial mesh



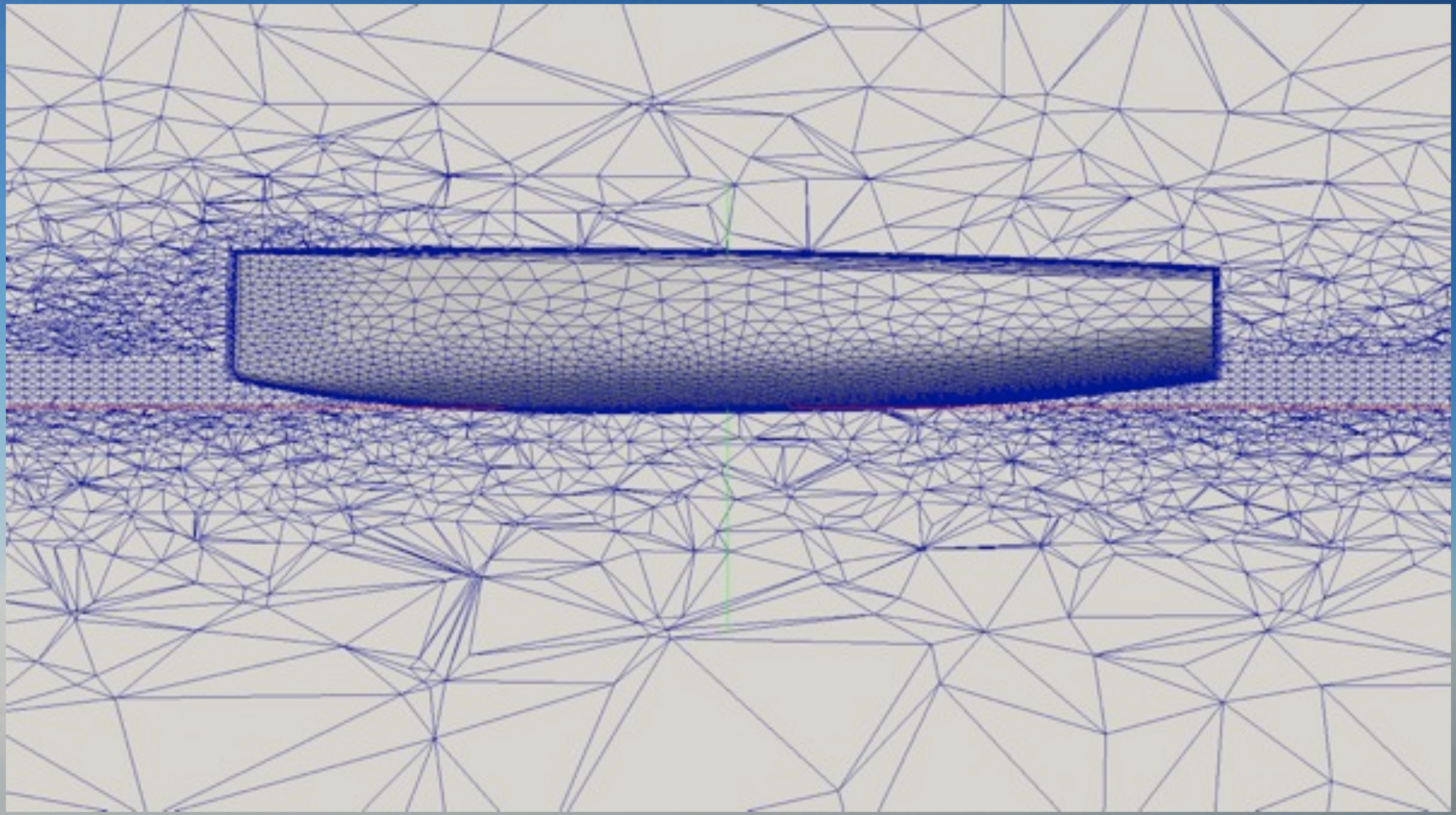
Deformed mesh at t = 2.25s. Rotation magnified 3 times



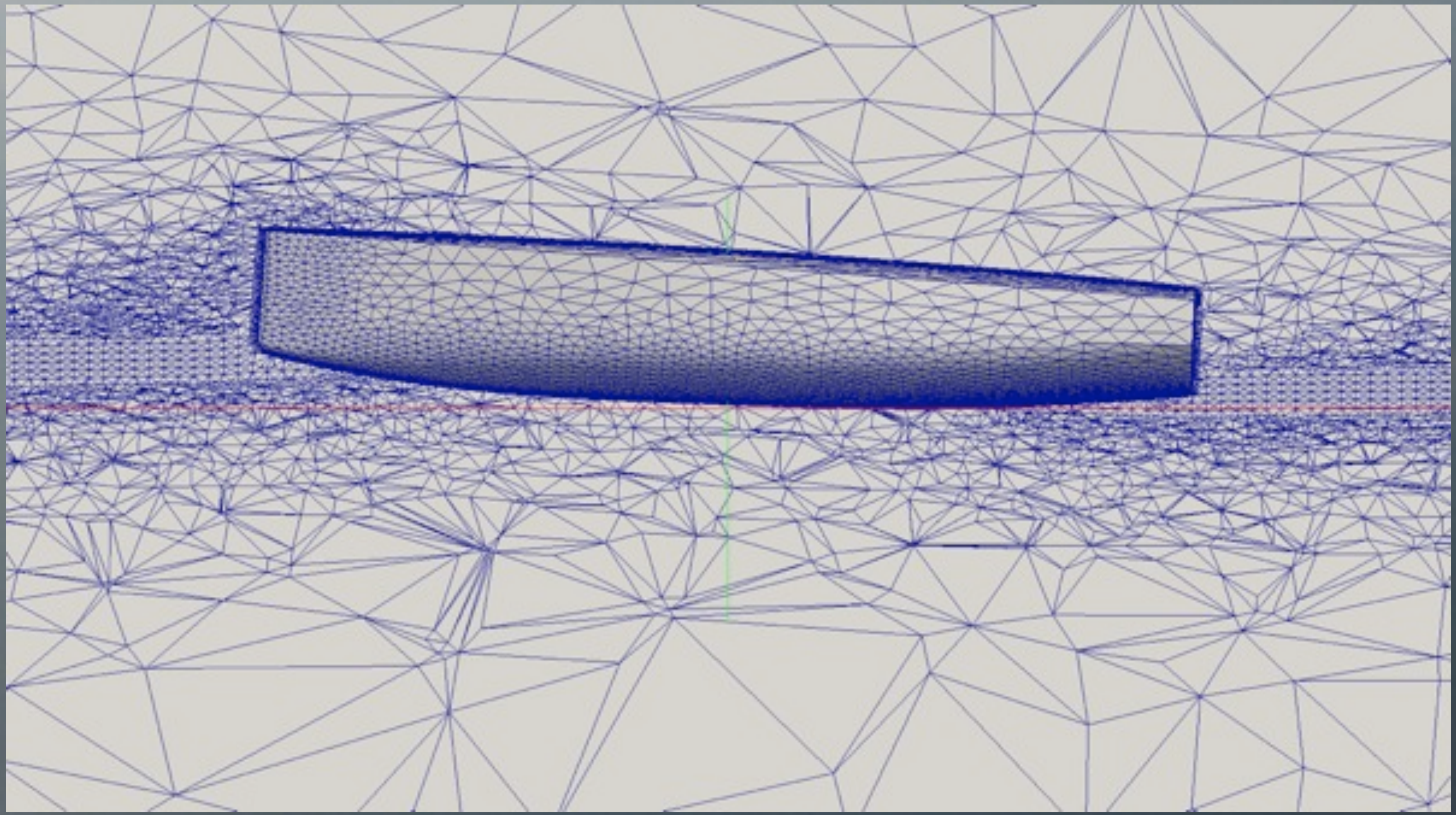
Race boat CFD

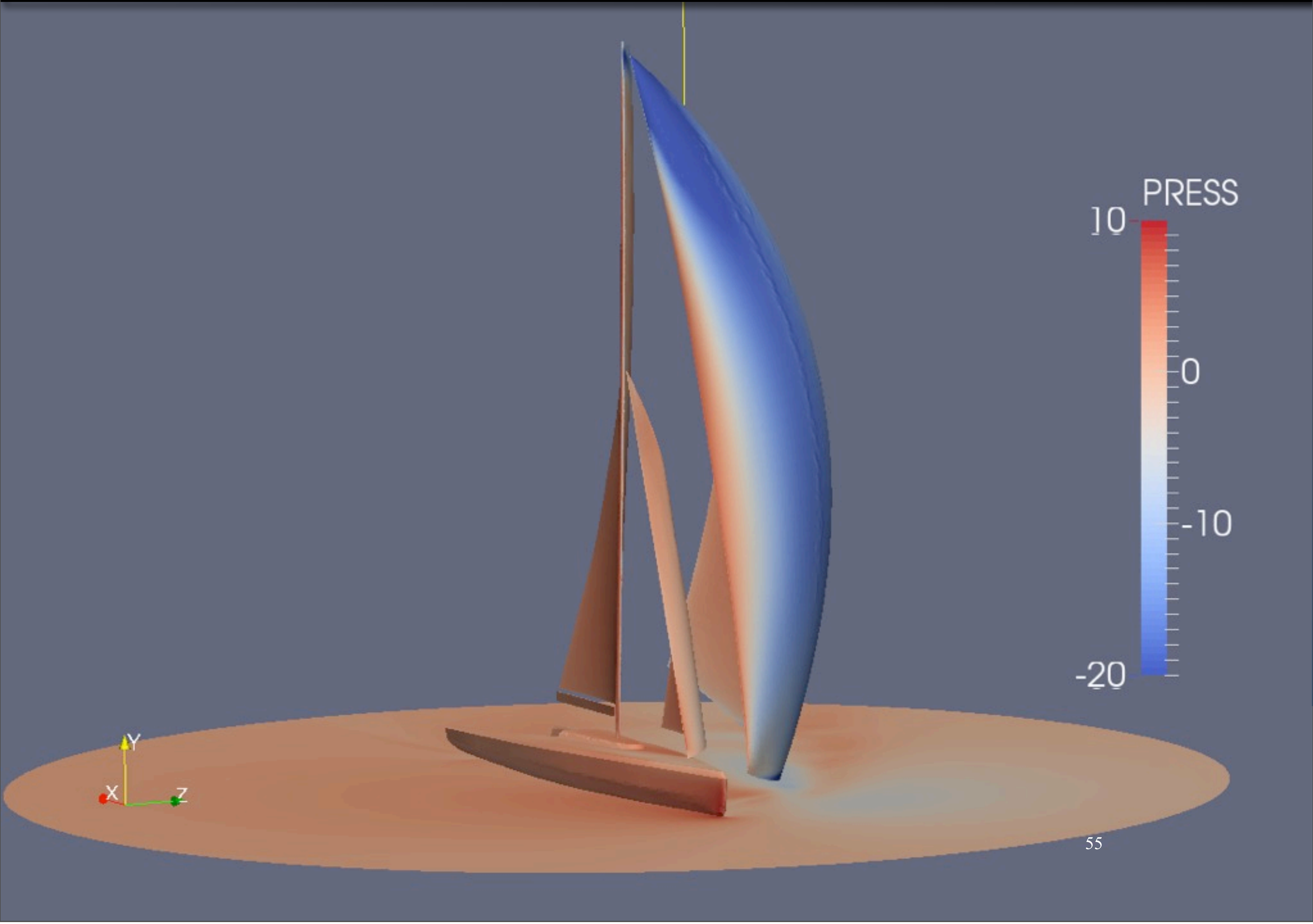
Race boat rigid body motion using ALE

Initial mesh

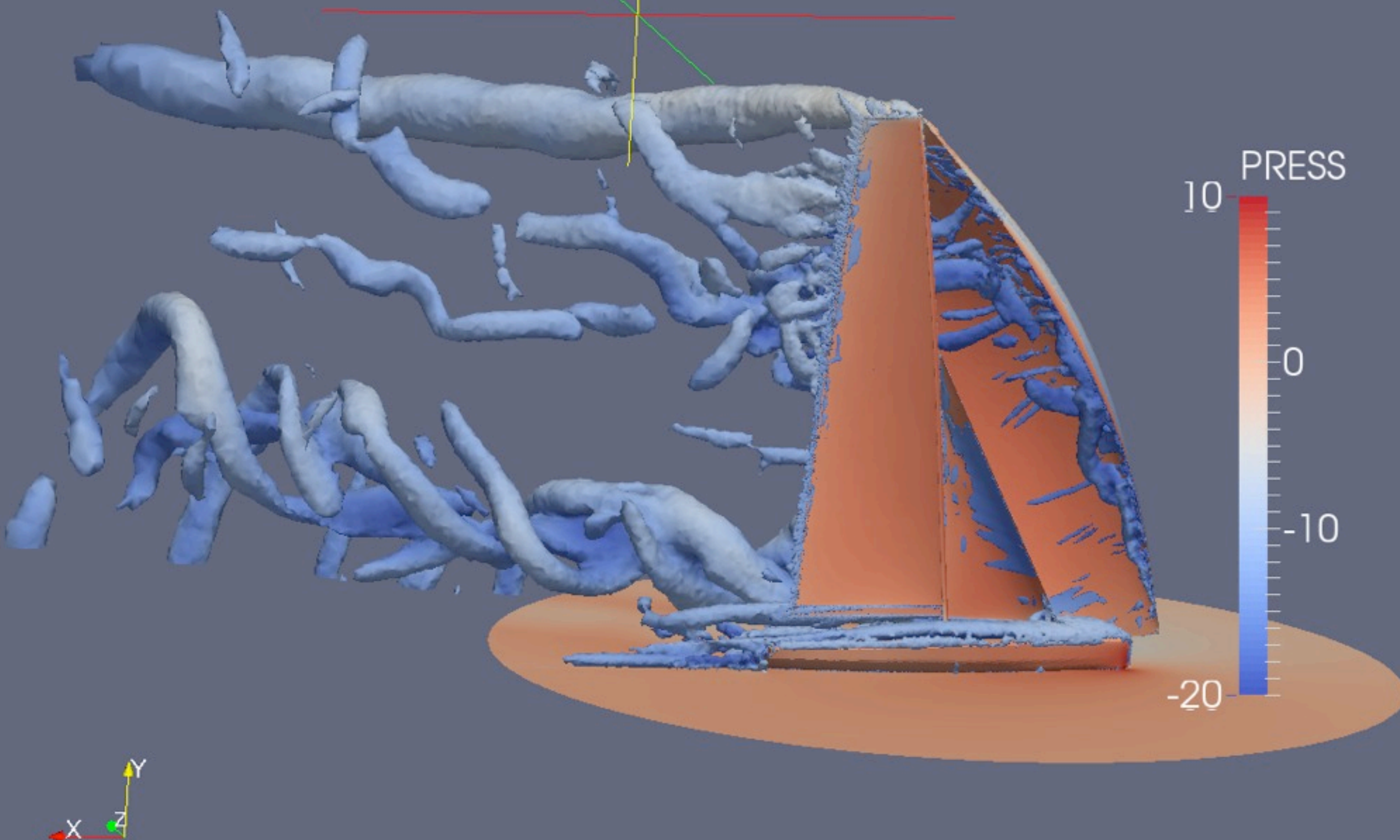


Deformed mesh at t = 6s.

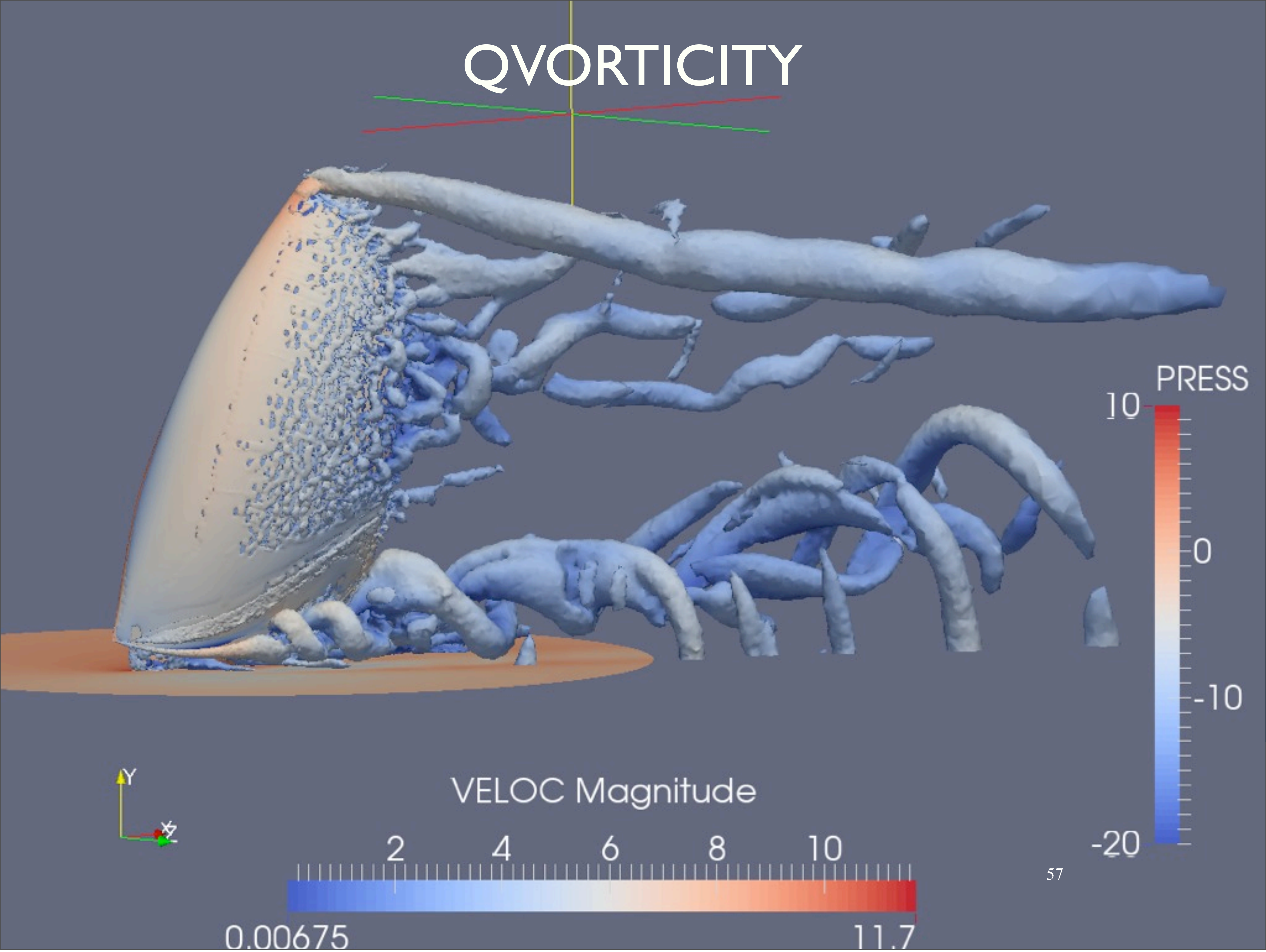



PRESS
10
0
-10
-20
Y
X
Z

QVORTICITY



QVORTICITY




Thanks!!!